



HEWLETT  
PACKARD

PC Interfacing

Owner's  
Guide

# HP 61016A Digitizing Oscilloscope



---

## Notice

The information in this document is subject to change without notice. This document contains proprietary information which is protected by copyright. All rights are reserved. No part of this document may be photocopied, reproduced, or translated to another program language without the prior written consent of Hewlett-Packard Company.

© 1985 by Hewlett-Packard Co.

---

## Printing History

New editions of this manual will incorporate all material since the previous edition. Update packages, which may be issued between editions, contain replacement and additional pages to be merged into the manual by the user.

The manual printing date and part number indicate its current edition. The printing date changes when a new edition is printed. (Minor corrections and updates which are incorporated at reprint do not cause the date to change.)

The instrument prefix number alongside the date refers to the first part of the serial number on the bottom of the instrument. This number indicates the version of the instrument that was available at the time that this manual was issued. However, note that, many instrument updates do not require manual changes and, conversely, manual corrections may be done without accompanying product changes. Therefore, do not expect a one-to-one correspondence between instrument changes and manual updates.

Edition 1....May 1985....Instrument Prefix 2514A



## Safety Summary

---

The following safety precautions must be observed during all phases of operation of this instrument. Failure to comply with these precautions or with specific warnings elsewhere in this manual violates safety standards of design, manufacture, and intended use of the instrument. Hewlett-Packard assumes no liability for the customer's failure to comply with these requirements.

### **Ground the Instrument**

To avoid potentially hazardous electrical shock, establish a safety ground before connecting user's circuits. Connect the output cable from the Power Pack to the Digitizing Oscilloscope, and then connect the line cord from the Power Pack to the ac line. The circuit or device that you are measuring must have an earth ground at its chassis or ground plane. *Do not attempt "floating" measurements with this instrument.* Detailed instructions are in the HP PC Instruments *System Owner's Guide* and in Chapter 4 of this guide.

### **Do Not Exceed Input Ratings**

Excessive input voltage and current will damage this instrument or its input probe. Do not exceed the "Maximum Input Voltage" specifications listed in Table 1-1 of this guide.

## Safety Symbols



Instruction guide symbol: the product will be marked with this symbol when it is necessary for you to refer to the guide (see What's in this Guide?)



Indicates measuring earth (ground) terminal.

**WARNING**

The WARNING sign calls attention to a procedure, practice, or the like, which, if not correctly performed or adhered to could result in personal injury. Do not proceed beyond a WARNING sign until the indicated conditions are fully understood and met.

**CAUTION**

The CAUTION sign calls attention to an operating procedure, or the like, which, if not correctly performed or adhered to could result in damage to or destruction of part or all of the product. Do not proceed beyond a CAUTION sign until the indicated conditions are fully understood and met.

## What's in this Guide?

---

This guide is a supplement to the HP PC Instruments System Owner's Guide and contains specific information for your Hewlett-Packard Digitizing Oscilloscope, Model 61016A. **You must read your System Owner's Guide before you read this guide!** Warranty and service information is included in the PC Instruments Support Guide, included with your System Owner's Guide.

The *System Owner's Guide* discusses information that is common to all HP PC instruments. It also contains specific information about your computer and the HP PC Instruments System. If you are a first time user, refer to Table 3 in your *System Owner's Guide* for the proper reading sequence of your computer and HP PC Instruments guides. Experienced users who already have an installed PC Instruments System need only read this guide to learn how to operate and program the Digitizing Oscilloscope.

You may insert this guide in the same hardcover binder as your *System Owner's Guide*. Here is a brief description of the contents of each chapter in this guide.

### **Chapter 1 - Product Description**

Briefly describes the Digitizing Oscilloscope, gives its specifications and general characteristics, and lists the items that you receive with it.

### **Chapter 2 - Trying Out Your Instrument**

Gives simple step-by-step instructions that let you quickly learn operations with nothing connected to the front panel.

### **Chapter 3 - Manual Instrument Control**

Gives detailed manual-control operating information not covered in the simplified instructions of Chapter 2.



#### **Chapter 4 - Front Panel Connections**

Explains how to select the proper probes and connect them to the Digitizing Oscilloscope and your application. Also included is a brief probe compensating procedure for divider probes.

#### **Chapter 5 - Programming with BASIC**

Explains how to control the Digitizing Oscilloscope with a computer program. All program statements for the Digitizing Oscilloscope are described. A simple programming example is also included.

#### **Appendix A - Programming Statement Summary**

Lists all programming statements that apply to the Digitizing Oscilloscope.

#### **Appendix B - Verification and Calibration**

Describes verification and calibration procedures that you can use to verify proper operation of the Digitizing Oscilloscope if you suspect an instrument malfunction.

#### **Appendix C - Error Messages**

Lists all error messages that apply to the Digitizing Oscilloscope.



# 1

## Product Description

### Introducing the HP PC Instruments Digitizing Oscilloscope

The HP PC Instruments Digitizing Oscilloscope is a 2-Channel, 50-MHz repetitive-bandwidth, digitizing oscilloscope that is fully programmable, self-calibrating, and capable of parametric measurements. It performs its functions under the control of your personal computer that is equipped with the HP PC Instruments software and the appropriate PC Instruments Interface Card. The *System Owner's Guide* explains the software and the interface card required for your computer. Chapters 2 and 3 of this guide explain how to operate the Digitizing Oscilloscope "manually" via the Soft Front Panel (computer display). Chapter 5 describes programming statements you can use for controlling the Digitizing Oscilloscope via the BASIC programming language.

Figure 1-1 is a simplified block diagram of the Digitizing Oscilloscope. Each vertical channel provides a 1M-ohm input with 10 sensitivity ranges from 5 mV/div to 5 V/div in 1- 2- 5 steps. The inputs can be a-c or d-c coupled. Each channel is also capable of offset.

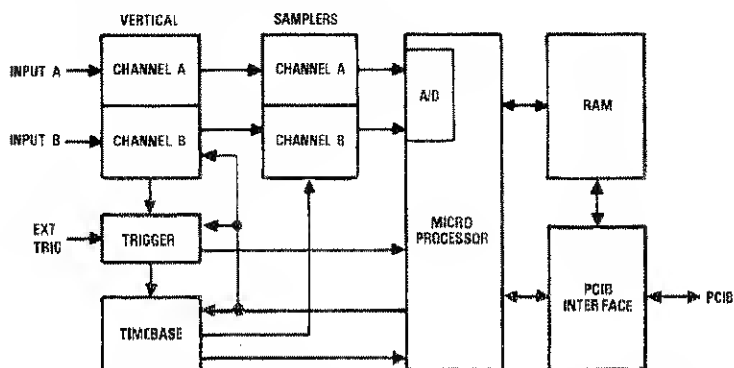


Figure 1-1. Digitizing Oscilloscope, Simplified Block Diagram

The timebase provides time ranges from 10 ns/div to 500 ms/div in a 1, 2, 5 sequence, with an adjustable delay of -1 to 250 screen diameters from the trigger point.

Triggering can be obtained from either Channel A or B (internal) or externally. The internal trigger range is adjustable over the entire vertical range and offset. The external trigger requires a 1-volt rising edge; input impedance is 100 k ohms.

Waveform data is acquired by random repetitive sampling at a sampling rate of 2 MHz (maximum) and an A/D conversion rate of 5 kHz (maximum).

---

## Items Supplied

In addition to this owner's guide, check that you have received the following items with your Digitizing Oscilloscope:

**Power Pack** - an a-c power transformer with an attached one metre cable. The transformer type you received was determined by the available a-c voltage in your country. Chapter 2 of the *System Owner's Guide* lists the different types and their HP part numbers.

**Power Cord** - connects the Power Pack to an a-c source. The plug type was determined by the country of destination. Chapter 2 of the *System Owner's Guide* lists types and HP part numbers.

**Instrument Interconnect Cable (HP 8120-4631)** - connects two stacked instruments together. Refer to the *System Owner's Guide* for installation instructions.

**Owner's Guide Update Pages** - if applicable, updated pages are included. Replace the obsolete pages with the new ones before you use this guide.

---

## Accessories

The following miniature probes, each with one-metre cable, are available for use with the Digitizing Oscilloscope:

**HP 10040A** - 10:1 division ratio and 9 pF shunt capacitance.

**HP 10021A** - 1:1 division ratio and 36 pF shunt capacitance.

---

## Specifications

Table 1-1 lists complete specifications for the HP 61016A Digitizing Oscilloscope equipped with a 1:1 input probe. All specifications apply over an operating temperature range of 0° C (32° F) to 40° C (104° F) after self-calibration.

Table 1-2 lists general characteristics, which are not specifications but typical values included for additional information.

Table 1-1. Specifications

**VERTICAL**

**Bandwidth (– 3 db):**

DC-coupled: dc to 50 MHz

AC-coupled: 10 Hz to 50 MHz

**Input Coupling:**

AC or DC

**Input Impedance:**

1 Megohm  $\pm 2\%$ , shunted by approximately 18 pF

**Maximum Input Voltage:**

*Oscilloscope alone*  $\pm 40$  V (dc + peak ac).

*With 1:1 probe* lesser of probe maximum safe voltage or  
 $\pm 40$  V (dc + peak ac)

*With 10: 1 probe* lesser of probe maximum safe voltage or  
 $\pm 400$  V (dc + peak ac)

**Vertical Range:**

40 mV to 40 V, full scale

**Sensitivity:**

5 mV/div to 5 V/div, in 1-2-5 steps

**Resolution:**

(Trigger level set within vertical range and offset set to zero).

<i>Vertical Range</i>	<i>Resolution</i>
40 mV to 80 mV	0.67 mV
160 mV to 40 V	range divided by 240

**Gain Accuracy:**

$\pm 3\%$ .

Table 1-1. Specifications (continued)

**Zero Offset Error:**

$\pm 3\%$  full scale,  $\pm 3$  mV.

**Offset Range:**

<i>Full-Scale Vertical Range</i>	<i>Offset Range</i>
40 mV to 4 V	$\pm(1.5 \times \text{range})$
8 V to 16 V	$\pm 12$ volts
40 V	(none permitted)

**TIMEBASE**

**Timebase Range:**

100 ns to 5 seconds, full scale

**Sweep Speed:**

10 ns/div to 500 ms/div, in 1-2-5 steps

**Resolution:**

<i>Timebase Range</i>	<i>Resolution</i>
100 ns to 200 ns	1 ns
500 ns to 5 s	range divided by 250

**Delay Range:**

$-0.5$  to  $250 \times$  timebase range, with trigger referenced to the center.

**TRIGGER**

**Source:**

Either channel (positive or negative slope), or external.

**Range:**

$\pm 2$  times the vertical range; limited to  $\pm 20$  V.

**Sensitivity:**

<i>Frequency</i>	<i>Vertical Range</i>	
	<i>40 mV to 1.6 V</i>	<i>4 V to 40 V</i>
<10 MHz	15 mV	400 mV
>10 MHz	40 mV	1 V

Table 1-1. Specifications (continued)

**Level Accuracy:**

<i>Vertical Range</i>	
<i>40 mV to 1.6 V</i>	<i>4 V to 40 V</i>
$\pm 3\%, \pm 10 \text{ mV}$	$\pm 3\%, \pm 250 \text{ mV}$

**External Trigger:**

1 volt rising edge into 100 k ohm, with a risetime  
< 1  $\mu\text{s}$ .

**FACTORY DEFAULT LABELS AND SETTINGS**

**Label:**

SCOPE.01

**Vertical:**

<i>Setting</i>	<i>Channel A</i>	<i>Channel B</i>
On/Off *	On	Off
Volts/division	200 mV/div	200 mV/div
Offset	0.0 V	0.0 V
Coupling	ac	ac
Polarity	positive	positive
Probe attenuation	$\times 1$	$\times 1$

**Trigger:**

*Level* 0.0 V    *Slope* positive    *Mode* triggered

**Timebase:**

*Time/Division:* 100  $\mu\text{s}/\text{div}$     *Delay* 0.0 s

**Trigger Source:** Channel A

**Table 1-1. Specifications (continued)**

**FACTORY DEFAULT LABELS AND SETTINGS (continued)**

**Display:\***

<b>Display mode</b>	Averaged	<b>Number of Averages</b>	1
<b>Graticule type</b>	Axis	<b>Number of Screens</b>	1

**Acquisition Timeout (Programmed Mode Only):**

1.6 seconds

\* These settings apply only to the manual (Soft Front Panel) mode of operation.

**Table 1-2. General Characteristics**

**VERTICAL**

**Offset Accuracy:**

Zero offset error + gain error.

**Noise:**

1.5% of full scale or 2.4 mV, whichever is larger.

**Single Marker Accuracy:**

Gain accuracy + zero offset error.

**Dual Marker Accuracy:**

Gain accuracy.

**Probe scaling factors:**

1:1 or 10:1.

**Probe compensation signal:**

Square wave, approximately 7 kHz, 500 mV p-p

**Table 1-2. General Characteristics (Continued)**

**TRIGGER**

**Modes:**

Triggered, Auto Trigger, Auto Level

Auto trigger generates internal triggers at 40 Hz rate in absence of an input trigger.

Auto level continually adjusts the trigger level to track the input signals with duty cycles between 30% and 70%.

**TIMEBASE**

**Delay Accuracy:**

$\pm 0.02\%$ ,  $\pm 0.4\%$  of timebase range ( $\pm 10$  ns)

**Single Marker Accuracy:**

Delay accuracy.

**Dual Marker Accuracy:**

$\pm 0.4\%$  of timebase range ( $\pm 2$  ns)

**DIGITIZER**

**A/D Resolution:**

8 bits

**Digitizing Technique:**

<i>Timebase Range</i>	<i>Digitizing</i>	<i>Digitizing Rate</i>
100 ns to 50 $\mu$ s	Random repetitive	(not applicable)
100 $\mu$ s to 20 ms	Random sequential	5.814 kHz
50 ms to 5 s	Real time	250 divided by timebase range



**Table 1-2. General Characteristics (Continued)**

**Throughput:**

<i>Range</i>	<i>Samples/Second</i>
100 ns	300
200 ns to 100 $\mu$ s	700
> 200 $\mu$ s to 50 ms	Increasing to 2500 @ 50 ms
> 100 ms to 5 s	250 divided by the range

**MEASUREMENTS**

**Markers:**

Provided for manual timing and voltage measurements

**Automated Measurements for:**

Frequency, Period, Risettime, Falltime, + Width,  
— Width, P-P Volts, Preshoot, and Overshoot.

**File Management:** Waveforms may be saved and recalled for comparison.

**DISPLAY**

**Variable Persistence Mode:**

Displays samples for an update period set by the user and then erases them. The number of updates can be varied or set to infinite.

**Average Mode:**

Provides a display of the average of many samples. The averaging runs continuously, and can be set to the following number of samples:

1, 2, 4, 8, 16, 32, 64, 128.

**Table 1-2. General Characteristics (Continued)**

**AUTOSCALE**

The Autoscale feature displays both channels with the proper vertical, trigger, and timebase settings. Coupling is set to AC, and delay to zero. Requirements are:

Frequency > 50 Hz  
Duty Cycle 20% to 80%  
Amplitude > 20 mV

**SELF CALIBRATION**

This feature calibrates the Vertical, Trigger, and Timebase circuits to specifications. Self calibration automatically occurs when the instrument is first turned on, and also can be requested by the user at any time. Self calibration time is typically less than 3 seconds.

# 2

## Trying Out Your Instrument

---

### Introduction

The following step-by-step procedures allow you to quickly learn simple instrument operations. They are especially suitable for first-time users who want to quickly become familiar with the basic operation of their Digitizing Oscilloscope (Oscilloscope). Chapter 3 contains detailed operating information that you can use once you have learned the basics covered in this chapter.

---

### The Procedure

Trying out your instrument consists of pointing to and selecting various interactive fields on the Soft Front Panel. Before you try out your Oscilloscope, you already should have:

- connected it to the PC Instruments Interface
- applied power
- loaded the computer operating system
- loaded the PC Instruments software and configured the system
- renamed (or erased) "HPSTATE.HPC" to return the Oscilloscope to its factory default settings
- loaded and run the Soft Front Panel, as explained in Chapter 3 of your *System Owner's Guide*.

---

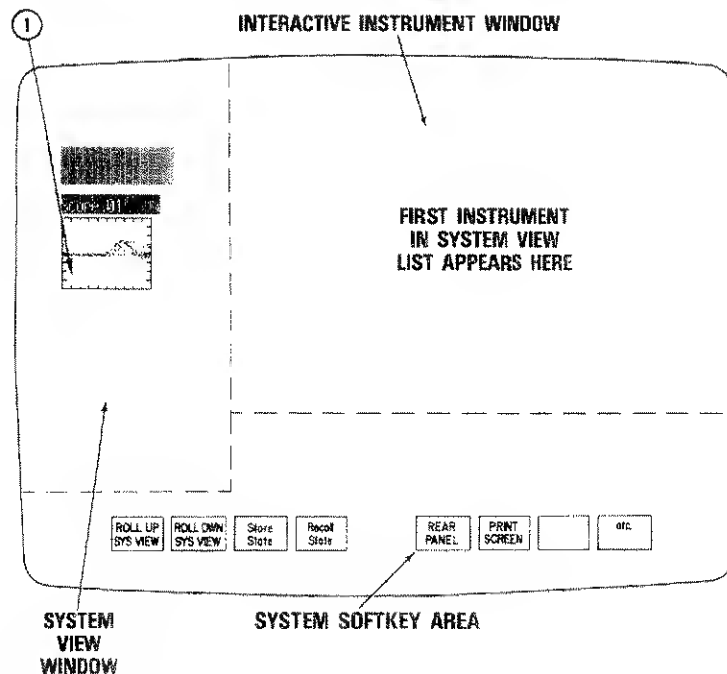
#### NOTE

*When you initialize the PC Instruments System with an Oscilloscope on the bus, there will be a delay of 5 to 20 seconds before the Soft Front Panel appears. This time is required to configure the various Oscilloscope functions.*

---

The following steps will guide you through the basic Soft Front Panel functions. Each step is keyed to an illustration that has a corresponding circled number.

**Step 1** - If the Oscilloscope is not already in the Interactive Instrument Window, point to and select SCOPE.01 from the labels listed in the System View Window (see Figure 2-1). If you have many instruments in your system, you may have to use the **ROLL UP SYS VIEW** and **ROLL DOWN SYS VIEW** softkeys to view a desired instrument. When you select an instrument from the window, the active indicator on the front of the instrument will light.



**Figure 2-1. Selecting the Instrument**

SCOPE.01 is the factory assigned default label for the first Oscilloscope in the system. If you are using more than one Oscilloscope in the system, each additional Oscilloscope is assigned a sequentially numbered default label (i.e., SCOPE.02, SCOPE.03, etc.).

## 2-2 Trying Out Your Instrument

**Step 2** - Once you select SCOPE.01, the SCOPE.01 area of the System View Window turns bright and your Oscilloscope appears in the Interactive Instrument Window (see Figure 2-2). That window has four fields unique to the Oscilloscope:

- Scopekeys
- Oscilloscope Status/Menu
- Oscilloscope Display
- Horizontal/Vertical Information

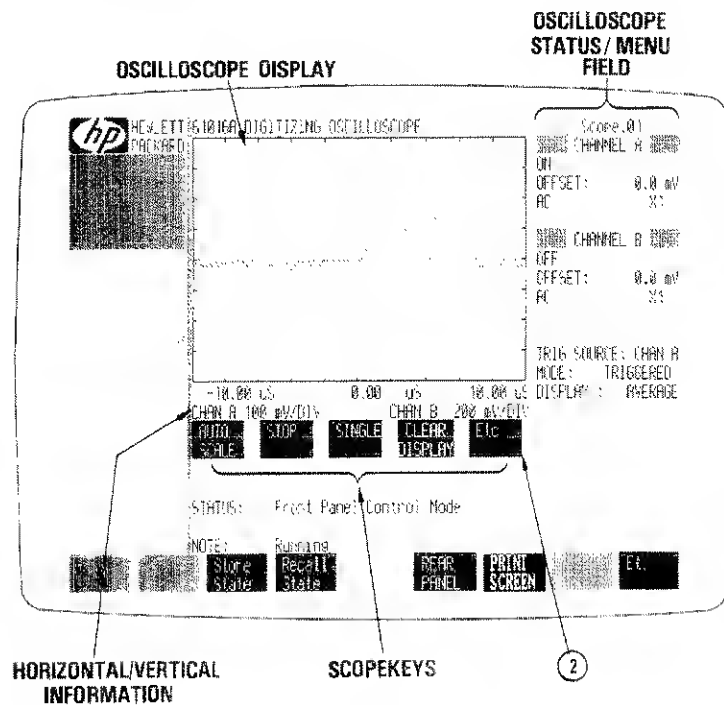
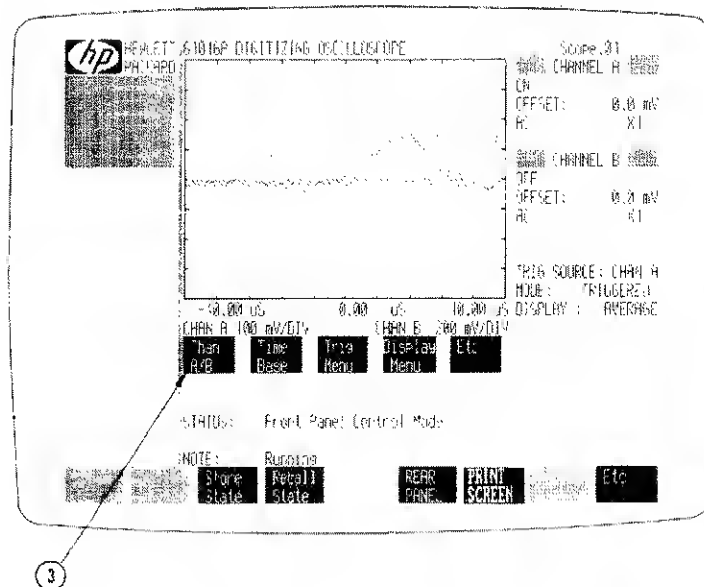


Figure 2-2. Oscilloscope, Front Panel View

The Scopekeys and Status/Menu fields let you control the Oscilloscope functions in the manual mode. The Scopekeys, **not to be confused with the System Menu Softkeys**, are specific to the Oscilloscope and allow you to conveniently operate the instrument's Soft Front Panel in the manual mode. Chapter 3 completely describes the Scopekeys and their functions.

There are four levels of Scopekeys. The default level, which is called level 1, is shown in Figure 2 -2. The level 1 Scopekeys control the major functions of the Oscilloscope. These functions, in contrast to those controlled by the remaining Scopekeys, do not display interactive menus but immediately execute the specified functions. To advance to next level of Scopekeys (level 2), select the Etc Scopekey once.

**Step 3.** The level 2 Scopekeys now appear in the Interactive Instrument Window (see Figure 2-3). These allow you to select the major Oscilloscope subsystems that must be properly configured to acquire and display a desired waveform. These Scopekeys give you access to interactive menu(s) displayed in the Oscilloscope Status/Menu field and allow you to make specific setup selections for your application. Point to and select the Chan A/B Scopekey to display the Channel A/Channel B Menu.



**Figure 2-3. Accessing a Level 2 Interactive Menu.**

**Step 4.** The Chan A/B Menu now appears in the Oscilloscope Status/Menu field of the Interactive Instrument Window (see Figure 2-4). Now, look at the bottom of the Oscilloscope Status/Menu field. That field has two interactive menu keys labeled **STATUS** and **Cont** (Continue). When **STATUS** and **Cont** are both displayed, more than one menu is available to you in that particular Scopekey group. Selecting **Cont** steps to the next menu in the selected group in essentially the same way that **Etc** steps through Scopekey levels.

If only the **STATUS** menu key appears, then there is only one menu in that group. Selecting **STATUS** causes the Oscilloscope Status/Menu field to exit the menu function and display the current Oscilloscope status. To return to the menu or menus in the same group, you must select the appropriate Scopekey for the desired menu group. To advance to the next scopekey level (level 3), select the **Etc** Scopekey again.

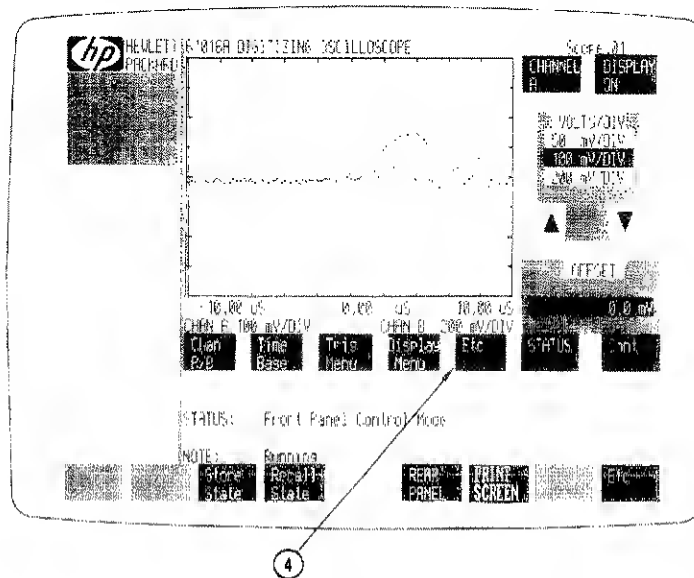


Figure 2-4. Advancing from Level 2 to Level 3



**Step 5.** The level 3 Scopekeys (see Figure 2-5) also give you access to interactive menu(s) that allow you to select specific measurements on the displayed waveform. For Example, with Wave Manage , you can save a waveform or recall a previously saved waveform for display. Select the Etc Scopekey once more to display the 4th, and last, Scopekey level.

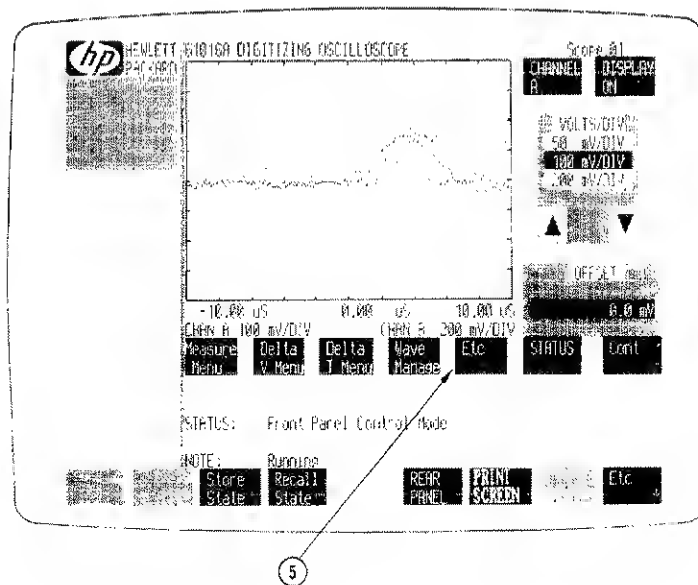
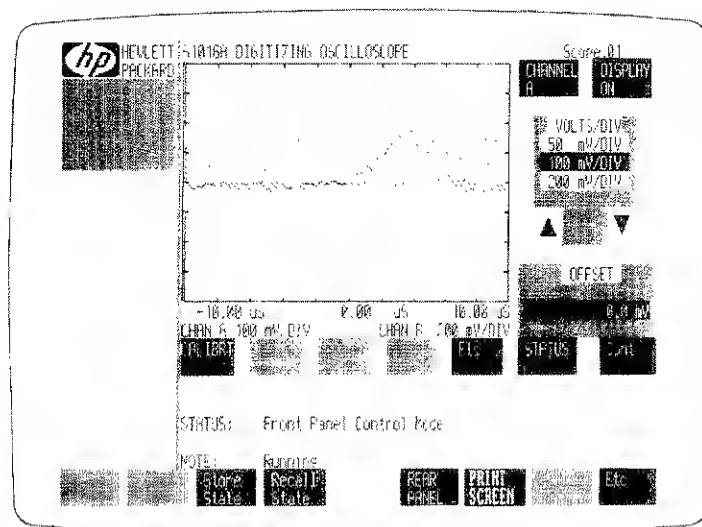


Figure 2-5. Advancing from Level 3 to Level 4

**Step 6** - You are now at the last level of Scopekeys (see Figure 2-6). By selecting CALIBRT you can start immediate execution of the self-calibration routine. When calibration is completed, the results are displayed.

Notice that you still have the Chan A/B Menu in the Menu/Status field. To choose another menu (Time Base, for example), select the Etc Scopekey twice. This advances you to the level 1 Scopekeys (Figure 2-2) and then to level 2 (Figure 2-3). Selecting the Etc Scopekey always advances you in the same direction; you cannot "back up" to the previous Scopekey level.



**Figure 2-6. Label 4 - The Calibration Scopekey**

---

## What to do Next

Now that you have familiarized yourself with the Oscilloscope controls via the Soft Front Panel, the next thing you do depends upon the type of user you are. If you are an experienced PC Instruments user and are already familiar with the *System Owner's Guide*, read the remaining chapters in this guide. If you are a first-time PC Instruments System user, read Chapter 4 in the *System Owner's Guide* and then Chapter 3 of this guide to learn about manual instrument control. Table 3 of the Introduction to the *System Owner's Guide* specifies the reading sequence of these manuals for first-time users.



# 3

## Manual Instrument Control

---

### Introduction

Chapter 2 gave you some basic information about using your Digitizing Oscilloscope (Oscilloscope) in the manual mode. Chapter 4 of the *System Owner's Guide* describes the features of the Soft Front Panel that are common to all instruments. This chapter discusses the Soft Front Panel controls that are specific to the Oscilloscope and, together with Chapter 4 of the *System Owner's Guide*, gives you the information needed to fully understand the manual operation of the Oscilloscope.

---

### Selecting the Instrument

To manually control or configure an instrument using the Soft Front Panel, you must point to and select its label in the System View Window. The Oscilloscope's default label is "SCOPE.01". Once you select its label, the Oscilloscope "front panel" appears in the Interactive Instrument Window. You use the front panel to operate the Oscilloscope's controls, which can be changed as often as you wish. If you then select a different instrument from the System View Window, the computer will "remember" the last control settings and they will determine how the Oscilloscope appears the next time you select it. You can change the selected instruments as many times as you wish without having to reconfigure them as long as you do not turn the computer off. The rest of this chapter tells you how to configure and operate the Oscilloscope.

---

## Operating from the Soft Front Panel

The Oscilloscope's front panel consists of four fields: Display, Oscilloscope Status/Menu, Scopekey, and Vertical/Horizontal Information. The Scopekey and Oscilloscope Status/Menu fields allow you to control the Oscilloscope, while the other two fields display information about the instrument's settings and the acquired waveform(s). Oscilloscope control is a bilevel process. The Scopekeys comprise the first level and the Oscilloscope Status/Menu is the second.

The first step in controlling the Oscilloscope is to select the appropriate Scopekey, which will either execute a complete operational function or display an interactive menu from which you complete a configuration or measurement setup. We will discuss the Scopekeys first, followed by the interactive menus.

---

## Scopekeys

Table 3-1 shows the Scopekeys, which are grouped into four functional levels. With the exception of the level 1 (Operation) Scopekeys and the Etc Scopekey, selecting a Scopekey displays a unique interactive menu in the Oscilloscope Status/Menu field, located in the upper-right portion of the Interactive Instrument Window.

Table 3-1. Scopekeys by Level and Function

Level	Function	Scopekeys				
1	Operation	AUTO SCALE	STOP (RUN)	SINGLE	CLEAR DISPLAY	Etc
2	Configuration	Chan A/B	Time Base	Trig Menu	Display Menu	Etc
3	Measurement	Measure Menu	Delta V Menu	Delta T Menu	Wave Manage	Etc
4	Calibration	CALIBRT				Etc

### Etc Scopekey (All levels)

The Etc Scopekey appears in each level. Use it to move to the desired Scopekey level. Each time you select Etc, you will be moved to the next level. For example, if the Operation Scopekeys (level 1) are displayed and you want to calibrate the instrument, select Etc three times. This displays the Calibration level, where the CALIBRT Scopekey resides. If you select Etc once more, you will be returned to the first level (Operation), not to level 3.

### Operation Scopekeys (Level 1)

These Scopekeys control the major operation of the Oscilloscope. When you select one of these Scopekeys, the appropriate function is executed without additional interaction on your part.



### AUTO SCALE

Automatically scales horizontal and vertical ranges, and sets the trigger in an attempt to display the waveform on screen. The trigger and horizontal ranges are determined by the signal on Channel A (if A is on, otherwise Channel B determines the settings). The vertical ranges are

determined for Channels A and B independently. Any settings you have previously made will be destroyed unless saved before initiating AUTOSCALE . AUTOSCALE overrides the STOP and SINGLE Scopekeys.

---

#### NOTE

*When you select AUTOSCALE , there is a 5 to 10-second delay before the Oscilloscope scaling is completed. This is normal and is due to the configuration complexity required by the Oscilloscope to perform its various functions.*

---

#### STOP (RUN)

Toggles between the RUN and STOP functions. In run mode, data acquisition and display start and the Scopekey shows STOP . To stop data acquisition and freeze the display, select STOP . The Scopekey legend changes to RUN . An advisory note, displayed in the System Status Window, indicates the Oscilloscope has been stopped. To restart data acquisition and display new data, select RUN . The Scopekey legend then changes back to STOP .

You may reconfigure the Oscilloscope while it is in STOP. However, data acquisition will not occur until you select RUN . The waveform, along with its supporting data structure, will be cleared.

#### SINGLE

Captures data for one trigger and adds it to the display. Data acquisition stops, the display is frozen, and the STOP Scopekey toggles to RUN . An advisory, displayed in the System Status Window, indicates the Oscilloscope is in single-mode. You can acquire and display additional data by selecting either SINGLE or RUN . Select SINGLE as many times as you want new data added to the display. To resume normal repetitive data acquisition, select RUN .



You may reconfigure the Oscilloscope in the single mode. If you select **SINGLE** again, new data is acquired and displayed. If you select **RUN**, the waveform and its supporting data structure is cleared and new data acquired and displayed.

**CLEAR  
DISPLAY**

Clears the displayed waveform and associated data structures. If the Oscilloscope is not stopped or in single mode, a new set of data is acquired and displayed each time you select **CLEAR DISPLAY**. All control settings (sweep speed, vertical sensitivity, etc.) remain unchanged.

**Configuration  
Scopekeys  
(Level 2)**

These Scopekeys control the major Oscilloscope front panel settings, such as vertical sensitivity, sweep speed, trigger source, etc. This group displays the appropriate selection menu(s) in the Oscilloscope Status/Menu field, from which you select specific setups.



**Chan  
A/B**

Allows access to the group of menus from which you choose Channel A and B settings. The choices are:

Channel A or B (On or Off); Vertical Sensitivity; Offset; Coupling; Polarity; Probe Attenuation

**Time  
Base**

Displays a menu from which you choose the time (horizontal sweep) settings. The choices are:

Time/Division; Delay

**Trig  
Menu**

Allows access to the group of menus from which you select the trigger parameters. The choices are:

Source; Level; Polarity; Mode

**Display  
Menu**

Allows access to the group of menus from which you choose the display parameters. The choices are:

Display Mode Type; Graticule Type; Number of Screens

### Measurement Scopekeys (Level 3)

This group of Scopekeys provide menus from which you choose the Oscilloscope's measurement functions. The menus appear in the Status/Menu field.



#### Measure Menu

Displays a menu that allows you to select a displayed waveform (either Channel A or Channel B) for parametric measurements. There are nine measurements; two typical ones are are:

Risetime; Peak-to-Peak Voltage

#### Delta V Menu

Allows access to the group of menus from which you specify the Delta V markers. These markers locate the points between which the change-of-voltage measurement is taken.

#### Delta T Menu

Allows access to the group of menus from which you select the Delta T markers. These markers locate the points between which the change-in-time measurement is taken.

#### Wave Manage

Allows you to save waveforms and critical scaling data for later review and to later recall them for waveform comparison.

### Calibration Scopekey (Level 4)

You will use this Scopekey level only when the Oscilloscope needs to be calibrated. The calibration is performed automatically by the software.



#### CALIBRT

When selected, this Scopekey provides a self-calibration of the Oscilloscope.

---

## NOTE

*When you select CALIBRT , there is a 5 to 20-second delay before the Oscilloscope calibration is completed. This is normal and is due to the complexity of the calibration procedure.*

---

---

## Menus

The rest of this chapter describes the organization of the Oscilloscope menus, how to locate a desired menu, and a detailed description of each menu.

### Introduction

The menus that allow you to control the Oscilloscope are displayed in the Oscilloscope Status/Menu field, located in the upper-right corner of the Interactive Instrument Window. The field has two modes: Oscilloscope Status and Oscilloscope Menu. In the Status mode, the field displays the instrument's status, such as which channels are on, offset, coupling, probe attenuation, etc. The Status mode is not interactive. In the Menu mode, the field displays the interactive menus that provide the second level of the bilevel instrument control process.

### Accessing a Menu

To gain access to the menus, select the appropriate Scopekey. Several Scopekeys allow you to access more than one menu; however, you can display only one menu at a time. You can quickly see if more than one menu is available in a Scopekey group by looking at the bottom of the menu field (see Figure 2-3). If there is a Cont Scopekey in the field, then there is more than one menu in that group. Otherwise, there is only one menu in the group. Selecting Cont steps to the next menu in the selected Scopekey group. Menu selection is limited to the selected group until you select another Scopekey or the STATUS scopekey.

The STATUS Scopekey appears at the bottom of all menu fields. When selected, it exits the menu function and displays the current Oscilloscope status in the field. To return to the menu or menus in the same group, select the appropriate Scopekey for that desired menu group. Then, if the desired menu is not the first in the group, select Cont .

### Finding the Desired Menu

In order to configure the Oscilloscope manually, you will need to find the desired menu. As previously discussed in "Scopekeys", the menus are grouped functionally. Table 3-2 is a quick-reference selection guide for finding the desired menu(s). Each menu is explained in detail in the "What's in Each Menu?" section. Figure 3-1 summarizes the menu selection process.

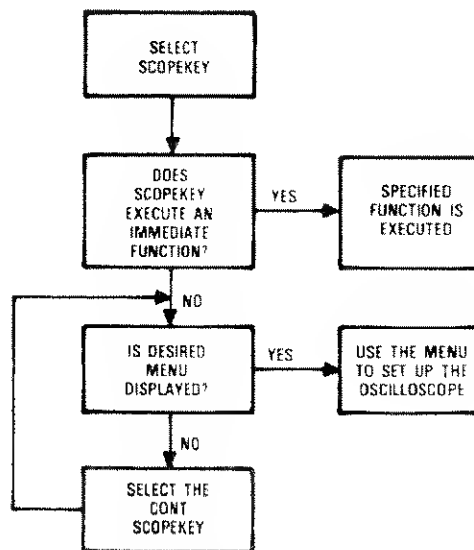


Figure 3-1. Menu Selection Flow Diagram

Table 3-2. Organization of the Oscilloscope Menus

Scopekey	No. of Menus	Menu 1	Menu 2
AUTO SCALE	*	These Scopekeys have no interactive menus and immediately execute a function. Refer to "Scopekeys" for details.	
STOP (RUN)	*		
SINGLE	*		
CLEAR DISPLAY	*		
Etc	*		
Chan A/B	2	channel A/B display ON/OFF volts/div offset	coupling trace polarity probe attenuation
Time Base	1	seconds/div delay	*
Trig Menu	2	source level	slope trigger mode
Display Menu	2	persistence/update time  or** average/no. of averages	graticule type full screen  split screen
Measure Menu	1	Channel A or Channel B  <b>NOTE</b> <i>The following waveform parameters are displayed for the specified channel as soon as it is selected:</i> risetime      falltime period      frequency + width      - width preshoot      overshoot p-p voltage	*

Table 3-2. Organization of the Oscilloscope Menus (cont.)

Scopekey	No. of Menus	Menu 1	Menu 2
Delta V Menu	2	marker select channel select markers on/off marker position marker movement rate	auto marker top/bottom auto marker position
Delta T Menu	2	marker select markers on/off marker position marker movement rate	channel select marker selection edge selection
Wave Manage	1	select channel to save specify record number specify record for recall	*
CALIBRT	1	No interactive menus. Self calibration executes immediately. Calibration results are displayed upon completion of the calibration routine.	

\* Not applicable.

\*\* Information in this menu depends on whether you select the Persistence or Average mode. Refer to "What's In Each Menu?" for details.

### Selecting the Desired Menu Function

We will refer to the interactive keys within each interactive menu as "switches". Two types of switches are described in the text; selector and toggle. The switch name describes how the menu key emulates a conventional switch. Depending on your type of computer, point to and select the appropriate switch by using the computer touchscreen, a "mouse" device, or the keyboard cursor keys.

### Making Numerical Entries

Unless otherwise specified, respond to menu requests for numerical entries with positive or negative real numbers expressed either in decimal or scientific notation.

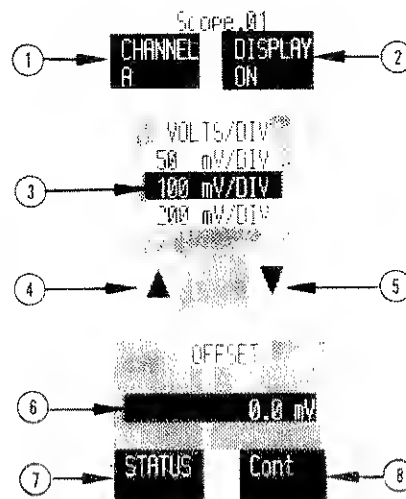
## What's In Each Menu?

This section describes the function of each menu in detail. In some cases, it may appear that there are more than two menus in a Scopekey group. When this occurs, the menu field actually displays a menu setup or state with no interactive keys except **Cont**, **STATUS**, or both. Such cases are noted in the specific menu description.

### Chan A/B Menu

The two channel menus allow you to set Oscilloscope functions generally associated with the vertical controls. You access these menus by selecting the **Chan A/B** Scopekey.

**Menu 1.** This menu allows you to set up the most frequently used vertical controls.



- ① Switch to toggle between CHANNEL A and CHANNEL B.
- ② Switch to toggle between channel DISPLAY ON and OFF.
- ③ To select a specific volts/division sensitivity range, use switches ④ ⑤ to step ranges up or down until the desired range appears in this window ③ . With desired range in the window, select the window to activate the new range. The active range is displayed in inverse video. The sensitivity ranges are:

(Millivolts) 5, 10, 20, 50, 100, 200, 500  
(Volts) 1, 2, 5

---

#### NOTE

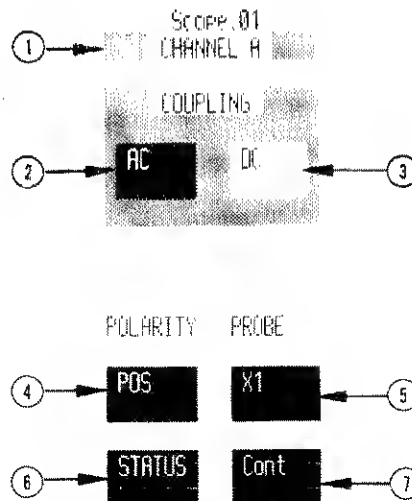
*If you are using a 10:1 probe, these ranges are multiplied by 10.*

---

- ④ ⑤ Switches to sequentially step the volts/div range up or down.
- ⑥ Use this window to set the vertical offset by entering a voltage value from the keyboard. Values are limited to 1.5 screen diameters (1 screen diameter = full scale), or 12 volts (multiply by 10 if your are using a 10:1 input probe). If you exceed the limit, an error message is displayed and the offset changed to the nearest in-range value.
- ⑦ Changes the field to display the Oscilloscope status.
- ⑧ Replaces this menu with Menu 2.



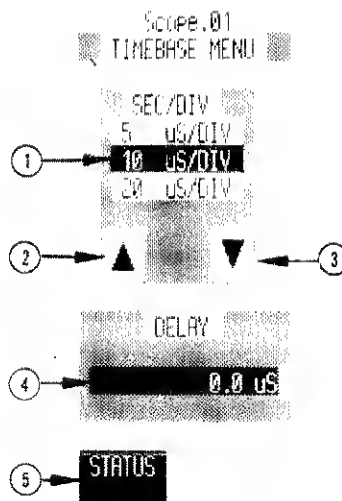
**Menu 2.** This menu allows you to set up the less frequently used vertical controls.



- ① Identifies the channel you are configuring.
- ② ③ Selector switches to set coupling to AC or DC.
- ④ Selector switch to toggle polarity between POS and NEG.
- ⑤ Switch to toggle probe attenuation between  $\times 1$  and  $\times 10$ . Amplitude values are displayed with this attenuation included for your convenience.
- ⑥ Changes the field to display the Oscilloscope status.
- ⑦ Replaces this menu with Menu 1.

### Time Base Menu

This menu allows you set the functions associated with the time (horizontal) controls of the Oscilloscope. The settings apply to one or both waveforms displayed in either a single or split-screen mode. You cannot set the time base range for each channel independently. Access this menu by selecting the Time Base Scopekey.



- ① To select the time/division range, use switches ② ③ to step ranges up or down until the desired range appears in this window ①. With the desired range in the window, select the window to activate the new range. The active range is displayed in inverse video. The sweep speed (time/div) ranges are:

(Nanoseconds ) 10,20,50,100,200,500

(Microseconds) 1,2,5,10,20,50,100,200,500

(Milliseconds) 1,2,5,10,20,50,100,200,500

- ② ③ Switches to sequentially step the time/div range up or down.
- ④ Use this window to enter the pre or post-trigger time value (use the keyboard to position the cursor and make the entry). Values must be within bounds of -0.5 to +250 screen diameters. If your value is out of bounds, an error message is displayed and the time reset to the nearest in-range value.
- ⑤ Changes field to display the Oscilloscope status.

## Trigger Menus

These menus allow you to set the functions associated with the trigger controls of the Oscilloscope. Access the menus by selecting the Trig Menu Scopekey.

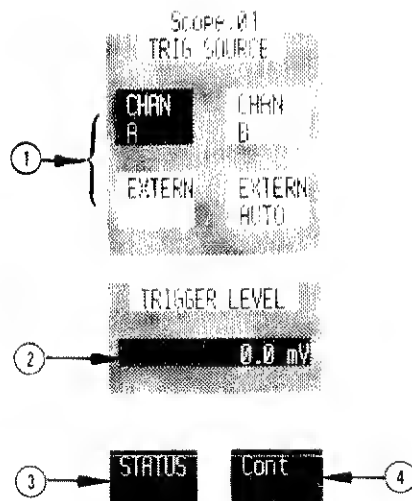
You may specify the trigger as CHAN A, CHAN B, EXTERN, or EXTERN AUTO. If you select CHAN A or B, you may then set slope, level, and trigger mode. If you select EXTERN as the source, the slope defaults to positive, level defaults to 1 V p-p, and mode defaults to triggered. If you select EXTERN AUTO, the Oscilloscope attempts to trigger on the same slope and level as external. However, if no trigger is found after 25 ms, the Oscilloscope triggers itself and acquires data.

If the trigger mode is set to TRIG or AUTO TRIG, you may set the slope and level. If the mode is set to AUTO LEVEL, you may set only the slope; the system sets and controls the level.

The triggered mode is the normal (default) mode of Oscilloscope operation. The Oscilloscope searches for a trigger of specified level and slope. If the proper level and slope are not found, the Oscilloscope does not trigger and no data is acquired.

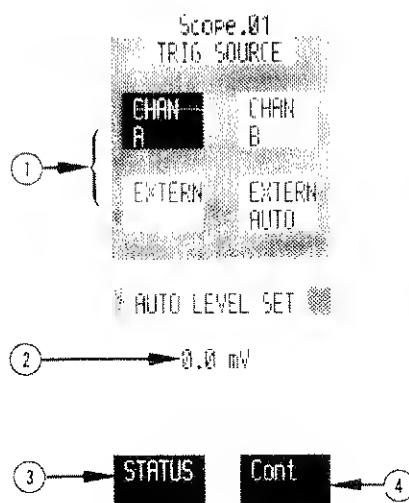
The AUTO TRIGGER mode allows the Oscilloscope to trigger on the level and slope that you set. However, if no trigger is found after 25 ms, the Oscilloscope triggers itself and acquires data. The AUTO LEVEL mode allows the Oscilloscope to maintain the trigger level based on the duty cycle of the waveform.

**Menu 1A.** This menu is displayed if the source is Channel A or B and the mode is TRIG(gered) or AUTO TRIG(ger). You can set the level from this menu and then go to Menu 2 to set the slope and mode.



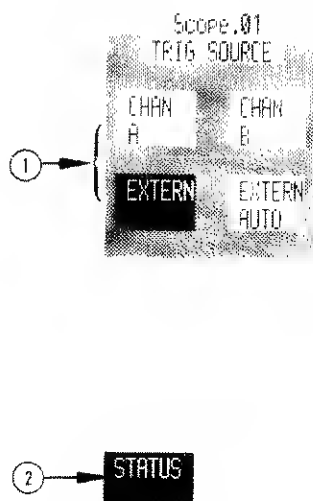
- ① Selector switch for setting the trigger source to CHAN A, CHAN B, EXTERN(al), or EXTERN(al) AUTO. If you select either external mode, Menu 1C is displayed.
- ② Use this window to set the trigger level by entering a volts or millivolts value (use the keyboard to position the cursor and make the entry). Values may be adjusted to twice the screen full scale, except for screen diameters of 16 V and 40 V, which are limited to 20 V (for a 1:1 input probe). If you exceed these limits, an error is displayed and the trigger level reset to the nearest in-range value.
- ③ Changes the field to display the Oscilloscope status.
- ④ Replaces this menu with Trigger Menu 2.

**Menu 1B.** This menu is displayed instead of Menu 1A when you set the trigger mode to AUTO LEVEL. Software and hardware continually monitor the trigger level, maintaining a stable trigger based on the duty cycle of the waveform. This menu then displays the level set by the system and you cannot change it.



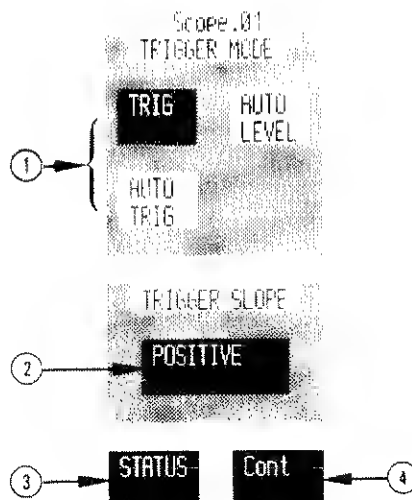
- ① Selector switches for setting the trigger source to CHAN A, CHAN B, EXTERN(al), or EXTERN(al) AUTO.
- ② Displays current trigger level set by hardware. This is not a numeric entry window if the mode is set to AUTO LEVEL.
- ③ Changes the field to display the Oscilloscope status.
- ④ Replaces this menu with Trigger Menu 2.

**Menu 1C.** This menu is displayed instead of Menu 1 when you select EXTERN or EXTERN AUTO as the trigger source. The slope and level are not displayed in EXTERN because they are controlled by hardware and cannot be changed. In EXTERN AUTO, the Oscilloscope attempts to trigger on the same slope and level as in EXTERN. However, if a trigger does not occur, the Oscilloscope triggers automatically after 25 ms.



- ① Switch for setting the trigger source to CHAN A, CHAN B, EXTERN(al), or EXTERN(al) AUTO.
- ② Changes field to display the Oscilloscope status.

**Menu 2.** This menu allows you to set the trigger slope and trigger mode. You can access this menu only if the trigger source is CHAN A or CHAN B.



- ① Selector switch for setting the trigger mode to TRIG(gered), AUTO TRIG(ger), or AUTO LEVEL.
- ② Toggle switch for alternating TRIGGER SLOPE between POSITIVE and NEGATIVE.
- ③ Changes field to display the Oscilloscope status.
- ④ If trigger mode is TRIG or AUTO TRIG, replaces this menu with Menu 1. If trigger mode is AUTO LEVEL, replaces this menu with Menu 1B.

## Display Menus

The display menus allow you to configure the way the acquired waveforms are displayed in the Oscilloscope Display field of the Interactive Instrument Window. There are two display modes available—Average and Persistence, and the Persistence mode can be either infinite or variable. In order to choose the best mode for displaying the data in your application, you will need to know how the system handles the data.

**Data Collection From the Oscilloscope.** The Oscilloscope continuously collects 251 data values for each channel. This data is collected from the Oscilloscope several times a second for analysis and display. Each of the 251 data values represents a dot, positioned along the X-axis in a time "bucket" and positioned along the Y-axis in an amplitude "bucket". The computer displays these dots in the Display field in either the Persistence or Averaged mode.

**Persistence Mode.** There are two types of persistence: infinite and variable. The display menus allow you to choose the persistence type and, with variable persistence, the number of updates.

1. **Infinite Persistence.** This is a simple storage mode that continually writes data points in the "dot memory" of the Display field. The display is erased only when you select the CLEAR DISPLAY Scopekey or reconfigure the Oscilloscope. In this mode, you can indefinitely monitor a waveform for its worst-case behavior. It is ideal for characterizing worst-case jitter, noise, or for investigating rare or intermittently occurring phenomena that are difficult or impossible to capture and display on conventional storage Oscilloscopes. Infinite Persistence can draw a complete "envelope" of a waveform.
2. **Variable Persistence.** The purpose of Variable Persistence Mode is to closely emulate a conventional nonstorage analog Oscilloscope and



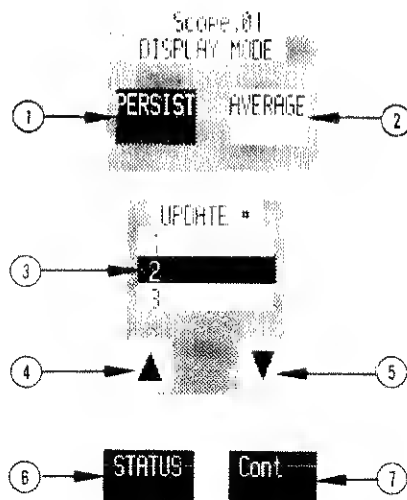
its ability to rapidly update the display. This allows you to see changes in the input signal as they occur.

The Variable Persistence Mode allows a limited number of unique dots on screen (2008 maximum per waveform). The dots remain on screen for the number of Oscilloscope "update cycles" that you specify in the display menus. An update cycle is the collection of a waveform data record from the Oscilloscope.

You may specify from 1 to 8 update cycles. Specifying 1 cycle clears the existing waveform, collects a new waveform from the Oscilloscope, and displays the new waveform data set. Specifying 8 update cycles maintains a record of 8 waveform data sets. Whenever the Oscilloscope collects a new waveform record, the oldest waveform record is erased, leaving the seven previous records and adding the new waveform to the display.

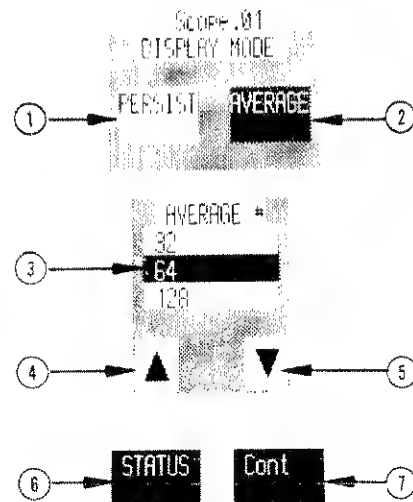
**Averaged Mode.** The Averaged Mode provides a statistical averaging of the data values for each time bucket. This mode has a smoothing effect on the display of the waveform by pulling the signal from the noise. The formula used by the software calculates a running average. The Display Menu allows you to select the number of averaged points.

**Menu 1A.** This menu is displayed when you have selected the Display Menu Scopekey and then the PERSIST(ence) Mode.



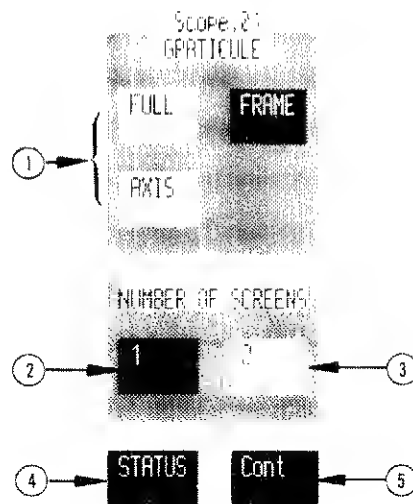
- ① ② Switches to set the mode to PERSIST(ence) or AVERAGE(d).
- ③ To select the number of updates, step the update choices up or down with switches ④ ⑤ until the desired number appears in this window ③. When the desired number is in the window, select the window to activate the new number of updates. The active number of updates is displayed in inverse video. The available choices are 1 through 8, and infinite.
- ④ ⑤ Switches to sequentially step the number of updates up or down.
- ⑥ Changes the field to display the Oscilloscope status.
- ⑦ Replaces this menu with Display Menu 2.

**Menu 1B.** This menu is displayed when you have selected the Display Menu Scopekey and then the AVERAGE Mode.

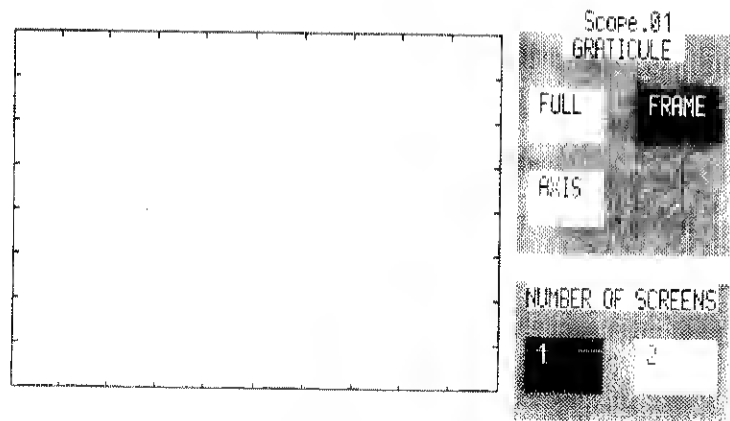


- ① ② Switches to select the PERSIST(ence) or AVERAGE(d) Mode.
- ③ To select the number of averages, first step the average choices up or down with switches ④ ⑤ until the desired number appears in this window ③. When the desired number is in the window, select the window to activate the new number of averages. The active number of averages is displayed in inverse video. Note that your available choices are powers of 2 from 1 to 128 (e.g. 1, 2, 4, 8, ..., 128).
- ④ ⑤ Switches to sequentially step the number of averages up or down.
- ⑥ Changes the field to display the Oscilloscope status.
- ⑦ Replaces this menu with Display Menu 2.

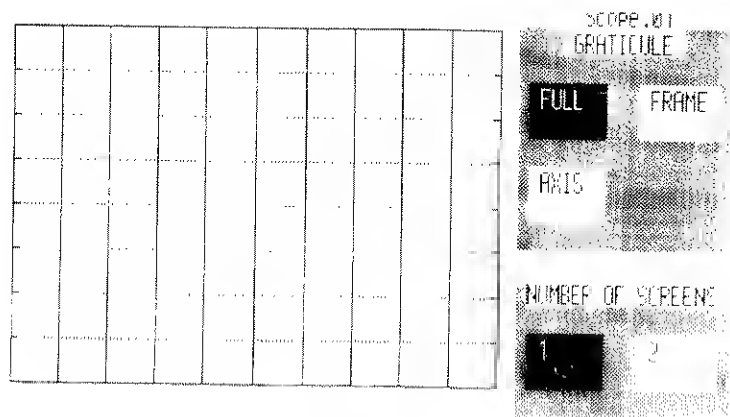
**Menu 2.** This menu allows you to select the graticule type and the number of screens. Figure 3-2 shows the types of graticules and number of screens available to you with this menu.



- ① Switches to set the graticule type to FULL, AXIS, or FRAME.
- ② ③ Switches to select the number of screens to appear in the Display field. The choices are 1 (full) or 2 (split) screen. If both channels are ON when you select split screen, Channel A will be displayed on top and Channel B on the bottom. However, you are not limited to a split screen when using two channels. If you choose full (1) when using two channels, both waveforms will appear on the same screen.
- ④ Changes the field to display the Oscilloscope status.
- ⑤ Replaces this menu with Menu 1A if you are in the PERSIST Mode, or to Menu 1B if you are in the AVERAGE Mode.

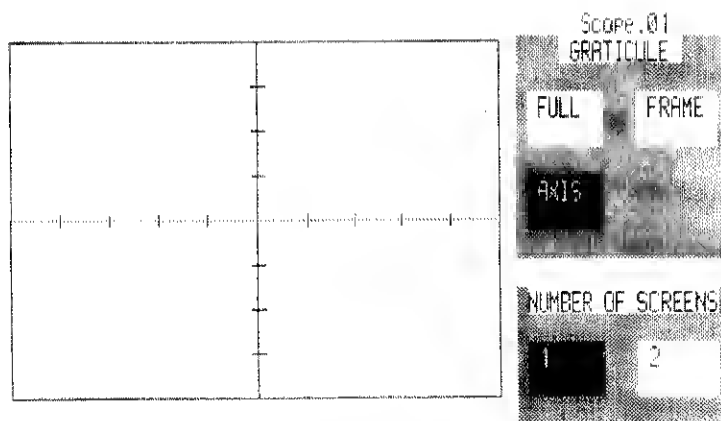


a) Frame

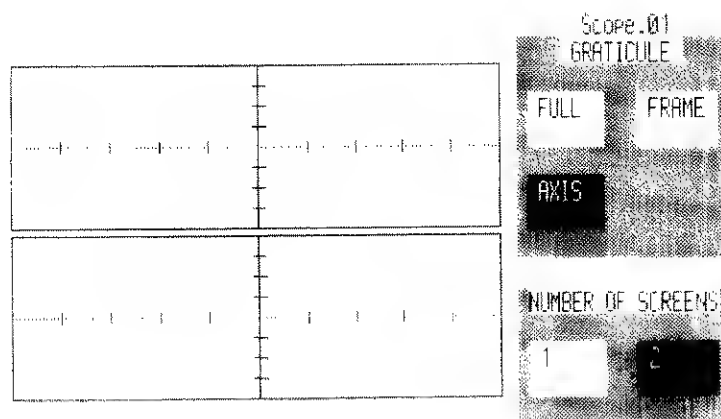


b) Full

Figure 3-2. Menu 2, Graticule and Screens



c) Axis



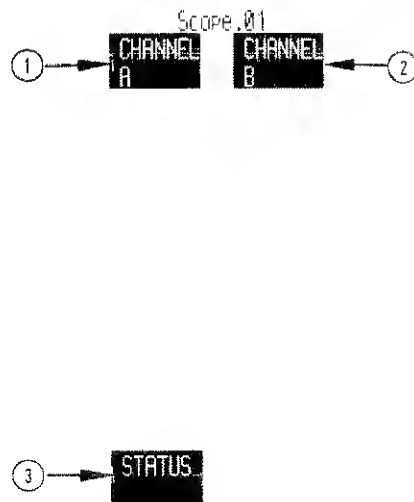
d) Split-Screen Axis

Figure 3-2. Menu 2. Graticule and Screens (cont.)

## Measure Menu

This menu allows you to select the channel on which automatic measurements are made and to view the results. When you select one of the momentary channel switches, the menu field automatically displays the measurement results for the waveform on that channel.

**Menu 1.** This menu allows you to select the channel for automatic measurements.



- ① ② Switches to select the measurement channel, which may be Channel A or Channel B.

---

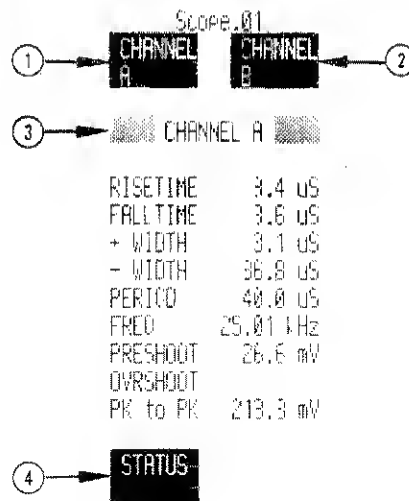
### NOTE

*When you select a channel for measurements, there is a 2 to 10-second delay before the measurement results are displayed. This is due to the number of calculations required to obtain the complete waveform analysis.*

---

- ③ Changes the field to display the Oscilloscope status.

**Menu 2.** After you make a selection from the first displays, this displays the results of the calculations.



#### NOTE

*The terms used in this field comply with standard IEEE definitions.*

- ① ② Switches to select the channel for which you want new measurement calculations.
- ③ Indicates the channel for which the measurement results are being displayed.
- ④ Changes the field to display the Oscilloscope status.



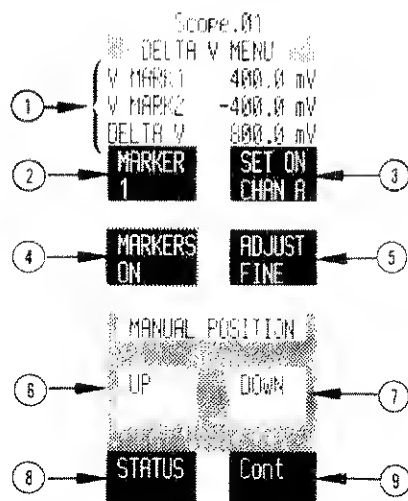
### **Delta V Menus**

These menus enable you to display and control two voltage markers (appearing as two horizontal lines) used for measuring absolute voltages or making differential voltage measurements. Use Menu 1 to position the markers manually or Menu 2 to position them automatically.

The **SET ON CHAN A/B** toggle switch allows you to select the channel to which the markers are referenced. For example, you may set Marker 1 to Channel A and Marker 2 to Channel B (Both channels must be on; you will get a message if a selected channel is turned off). If you assign markers to both channels, the DELTA V measurement represents the difference between the channel assigned Marker 1 and the channel assigned Marker 2. You cannot position the markers automatically unless they are both assigned to the same channel.

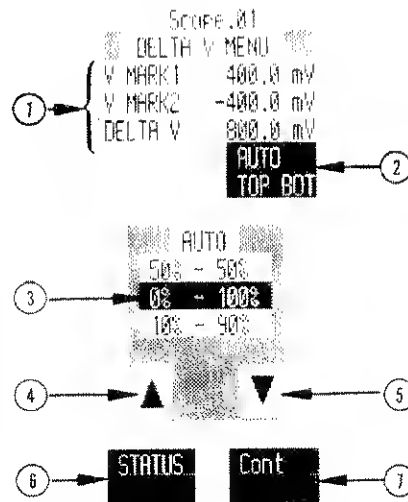
When the markers are on, the voltage of each marker and the difference voltage (DELTA V) between Marker 1 and Marker 2 are displayed in the menu field. When you leave this menu, DELTA V (only) is displayed in the Oscilloscope Status field until you turn the markers off. If you leave this menu and then wish to turn the markers off, you may use the **AUTOSCALE** Scopekey to do so. Or, you may return to this menu and use the Marker toggle switch.

**Menu 1.** This menu allows you to manually position markers for one or both channels. When the markers are OFF, only switch ④ is displayed. Once you select markers ON, the entire menu is displayed.



- ① Displays the voltages of Marker 1 (VMARK1) and Marker 2 (VMARK2) and difference of the markers as DELTA V.
- ② Toggle switch to select either Marker 1 or Marker 2.
- ③ Toggle switch to assign either Channel A or Channel B to the marker you selected with switch ②.
- ④ Toggle switch to turn the markers ON or OFF.
- ⑤ Toggle switch to select COARSE or FINE movement of markers. Markers move in large (COARSE) or small (FINE) increments when step switches ⑥ ⑦ are selected.
- ⑥ ⑦ Step switches to position the marker UP or DOWN.
- ⑧ Changes the field to display the Oscilloscope status.
- ⑨ Replaces this menu with Delta V Menu 2.

**Menu 2.** Use this menu to auto position the markers for a single-channel DELTA V measurement. Both markers *must* be assigned to the same channel.



- ① Displays the voltages of Marker 1 and Marker 2, and difference of Markers 1 and 2 as DELTA V.
- ② When you select this switch, the system finds the statistical top and bottom of the waveform. If AUTO position is selected, the marker positions are relative to the statistical top and bottom of the waveform. Once selected, the AUTO top and bottom values remain until the markers are positioned manually or turned off via Menu 1. If you do not select this switch, the top and bottom will default to the marker positions that you set manually in Menu 1.

---

### NOTE

*When you select AUTO position of the markers, there is a 2 to 10-second delay before measurement results are displayed. This is due to the number of calculations required to find the statistical top and bottom of the waveform.*

---

- ③ To select the automatic position points, step the percentage choices up or down with switches ④ ⑤ until the desired value appears in the window. When the desired value is in the window, select the window to activate the new position. The active position is displayed in inverse video. The AUTO position choices are:

0-100%, 10-90%, 20-80%, 50-50%,

- ④ ⑤ Switches to sequentially step the position choices up or down.  
⑥ Changes the field to display the Oscilloscope status.  
⑦ Replaces this menu with Menu 1.

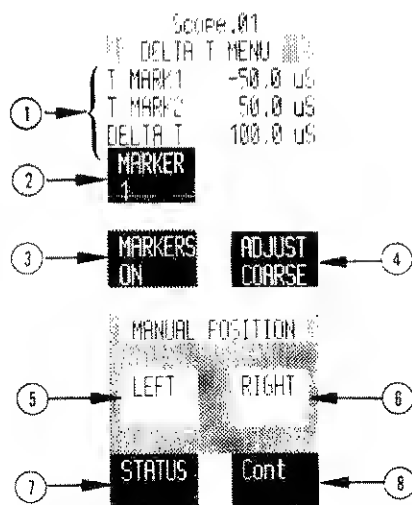
### Delta T Menus

These menus enable you to display and control two time markers (appearing as two vertical lines), used for measuring the time between the markers.

The Delta T markers are positioned relative to the timebase, which makes them independent of the number of channels that are ON. You may set the markers manually via Menu 1 or automatically with Menu 2. When you are using Menu 2 to set the markers automatically, they are positioned on the 50% points of the waveform.

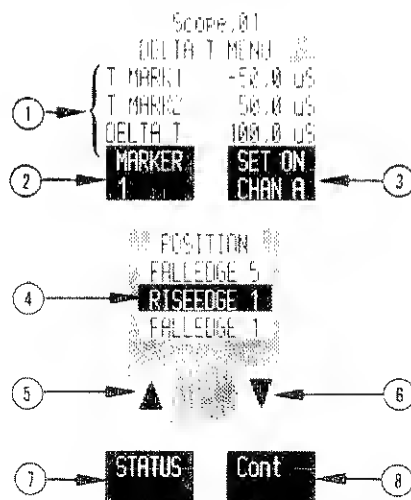
Once the markers are positioned, either manually or automatically, the time associated with each marker and the difference in time are displayed in the menu field. When you leave this menu, DELTA T is displayed in the Oscilloscope status field. If you leave this menu and then wish to turn the markers off, you may use the AUTO SCALE Scopekey to do so. Or, you may return to this menu and use the Marker toggle switch.

**Menu 1.** This menu allows you to manually position the two markers. When the markers are OFF, only switch ③ is displayed. Once you select markers ON, the entire menu is displayed.



- ① Displays the times of Marker 1 (TMARK1) and Marker 2 (TMARK2) with respect to the trigger point, and time difference between the markers as DELTA T.
- ② Toggle switch to specify MARKER 1 or MARKER 2.
- ③ Toggle switch to turn markers ON or OFF.
- ④ Toggle switch to select COARSE or FINE movement of markers. Markers move in large (COARSE) or small (FINE) increments when step switches ⑤ ⑥ are selected.
- ⑤ ⑥ Step switches to position markers LEFT or RIGHT.
- ⑦ Changes the field to display the Oscilloscope status.
- ⑧ Replaces this menu with Menu 2.

**Menu 2.** This menu allows you to automatically position markers on specified rising and falling edges of the waveform on a single channel.



- ① Displays the times of Marker 1 (T MARK1) and Marker 2 (T MARK2), and time difference between the markers as DELTA T.
- ② Toggle switch to select either MARKER 1 or MARKER 2.
- ③ Toggle switch to assign either Channel A or Channel B to the marker you select with switch ②.
- ④ To select the automatic position points, step the position choices up or down with switches ⑤ ⑥ until the desired choice appears in the window. Then, select the window to activate the new position. The active position is displayed in inverse video. The marker position choices are:

RISING EDGE 1, 2, 3, 4, or 5  
 FALLING EDGE 1, 2, 3, 4, or 5

---

### NOTE

*When you select AUTO position of the markers, there is a 2 to 10-second delay before measurement results are displayed. This is due to the number of calculations required to find the 50% points of the waveform.*

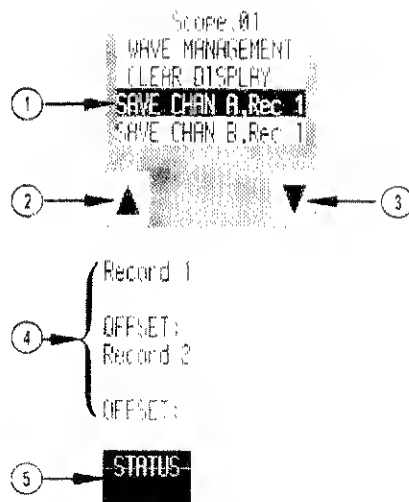
---

- ⑤ ⑥ Switches to sequentially step the marker position point up or down.
- ⑦ Changes the field to display the Oscilloscope status.
- ⑧ Replaces this menu with Menu 1.

### Wave Manage Menu

This menu allows you to save and recall the basic scale factors and waveform data of two different waveforms. When you save a waveform, the most recently displayed data is stored in a 251-byte array. In addition to the waveform data, the vertical, horizontal, and offset scale factors are also saved.

You may recall the saved waveforms for display or for comparisons with currently displayed waveforms. When the saved data is recalled, the waveform is displayed on the graticule and the scale factors are displayed in the status window.



- ① To select the desired WAVE MANAGEMENT option, step the choices up or down with switches ② ③ until the desired choice appears in the window. When the desired option is in the window, select the window to activate the new option. The active option is displayed in inverse video. The WAVE MANAGEMENT options are:

SAVE CHAN A, REC 1 (record 1)  
SAVE CHAN B, REC 1 (record 1)  
SAVE CHAN A, REC 2 (record 2)  
SAVE CHAN B, REC 2 (record 2)  
RECALL WAVE 1  
RECALL WAVE 2  
DISPOSE RECORD 1  
DISPOSE RECORD 2  
CLEAR DISPLAY

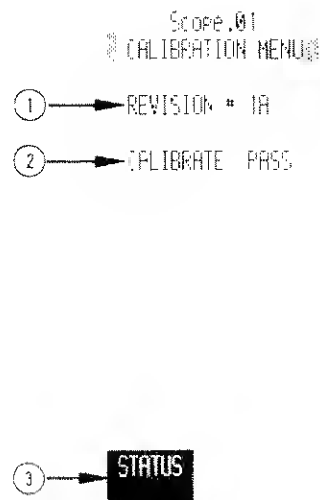
- ② ③ Switches to sequentially step the WAVE MANAGEMENT choices up or down.  
④ Displays the vertical sensitivity, sweep speed, and offset of each recalled record.  
⑤ Changes the field to display the Oscilloscope status.



## Calibration Menu

Selecting the CALIBRT Scopekey immediately initiates a self-calibration and all waveform data being acquired is lost. If the calibration is successful, the Oscilloscope Status/Menu field displays CALIBRATION PASS and data acquisition and waveform display is automatically restarted. If the calibration is unsuccessful, the field displays CALIBRATION FAIL. This indicates a hardware failure that must be corrected before you can use the Oscilloscope with any confidence.

While the Oscilloscope calibration routine is running, all other Scopekeys are disabled. They are re-enabled when the routine is finished.



- ① Oscilloscope firmware revision number (to be used by service personnel if needed. This number may differ from the one shown in this guide).
- ② Calibration PASS/FAIL message area.
- ③ Changes the field to display the Oscilloscope status.



# 4

## Front Panel Connections

---

### Introduction



This chapter describes how to connect the Digitizing Oscilloscope (Oscilloscope) to your application. It also describes the two types of accessory divider probes that you can use with the Oscilloscope. You should become familiar with the operation of the Oscilloscope, as explained in the two preceding chapters, before you connect anything to its front panel. You may also want to read about programming your instrument (in Chapter 5) before you make front panel connections.

---

#### **WARNING**

*Before making any front panel connections, establish a safety ground connection to your Oscilloscope by installing the approved Power Pack as described in the **System Owner's Guide**. The outer sleeve of each Oscilloscope BNC input connector is wired to earth ground through its Power Pack. The circuit or device that you are measuring must have an earth ground at its chassis or ground plane. **Do not attempt "floating" measurements.** Be certain that the cable shield of your Oscilloscope probe connects the BNC outer sleeve to the earth ground of the device you are measuring.*

---

---

## Application Connections

Before connecting the Oscilloscope to your application, you must decide on the type of input probe to use.

### Choice of Input Probe

The Oscilloscope and its software allow you to use either a 1:1 or 10:1 probe capable of matching the input impedance of the Oscilloscope, which is 1 megohm shunted by 18 picofarads. The choice of probe depends on the voltage and impedance of the source being measured. For example, you can use a 1:1 probe on low-voltage inputs as long as the input impedance of the Oscilloscope does not cause significant resistive and capacitive loading on the source you are measuring.

Use a 10:1 probe when the voltage being measured is high enough to require it, or if the 1:1 probe creates excessive resistive and/or capacitive loading. The Oscilloscope is designed to use divider probes having a 10:1 ratio. 10:1 divider probes are quite versatile because you can choose different probe input impedances to reduce circuit loading and still match the input impedance of the Oscilloscope.

### Calibrating the 10:1 Divider Probe

Before it is used, a 10:1 divider probe must be compensated for the variations in input impedance that occur between probes and Oscilloscopes.

---

#### NOTE

*Refer to your probe documentation for the specific procedure for, and location of, probe compensation adjustments.*

---

The Oscilloscope front panel provides a probe compensation signal (PROBE COMP) for compensating probes. PROBE COMP is a 500 mV p-p, 7-kHz square wave located on a "loop" type test point just inside the signal access hole. When the probe is properly

compensated, PROBE COMP (as viewed on the Oscilloscope through the 10:1 probe) will be a clean square wave with minimum overshoot or undershoot. Before performing the probe compensation adjustment, make the following changes to the oscilloscope's factory default configuration:

1. Change Channel A Probe Compensation from  $\times 1$  to  $\times 10$ .
2. Change Channel A Sensitivity from 2 V/div to 200 mV/div.
3. Change Time/division from 100  $\mu$ s/div to 50  $\mu$ s/div.

Connect the 10:1 probe from Channel A to the PROB COMP loop test point and adjust the probe for best square-wave response.

---

#### NOTE

*Once you have adjusted your probe for Channel A, it is matched only to that channel. To use the probe on Channel B, you must repeat the same compensation procedure on Channel B. If you use a separate 10:1 probe for each channel, you must compensate each probe for its respective channel.*

---

---

## Making Measurements

In addition to the measurement considerations already described, consider the following factors before making measurements:

- maximum input voltage capabilities of the probe/Oscilloscope combination.
- probe scaling factor
- warm-up period

### **Maximum Input Voltage**

The maximum safe input voltage for the Oscilloscope is  $\pm 40$  V (dc + peak ac). If the probe you are using has a lower maximum safe input voltage rating, use the probe rating as the maximum safe input voltage. Consult the documentation provided with your probe for its maximum safe input voltage.

---

#### **CAUTION**

The maximum safe input voltage for the Oscilloscope/1:1 probe combination is the lesser of the two maximum safe input voltages. The maximum safe input voltage for the Oscilloscope/10:1 probe combination is the lesser of the probe's maximum safe input voltage or  $\pm 400$  V (dc + peak ac). Exceeding these maximum safe input voltages will damage the probe and/or Oscilloscope.

---

### **Probe Scaling Factor**

The Oscilloscope can be configured for either a 1:1 or 10:1 probe. Once the probe scaling factor is properly configured, the waveform calculations automatically reflect the scaling factor. This frees you from having to make multiplications or divisions to determine the correct result.

### **Warm-Up Period**

The Oscilloscope specifications apply over an operating temperature of 0 to 40 °C **after** self calibration. To insure accurate measurements, allow the Oscilloscope to warm up for 15 minutes before performing the self calibration.

# 5

## Programming With BASIC

---

### Introduction

With the statements described in this chapter, you can write BASIC programs to:

- configure the Digitizing Oscilloscope (Oscilloscope)
- acquire waveform data
- perform waveform measurements
- display the results on the computer.
- program the Oscilloscope for automated tests and measurements.

Before you attempt this, you should be familiar with controlling the instrument in the manual mode (Chapters 2 and 3). Also, you must already know how to write programs in BASIC before you can write your own application program. Chapter 5 of the *System Owner's Guide* gives information about how to develop and run your program. Before writing your program, you must run the Soft Front Panel to:

- assign a label to the Oscilloscope
- create one or more State files
- generate a starter program (Program Shell)

This chapter describes all the statements that you can use in your program to control the Oscilloscope. These statements fall into two categories— system and instrument. System statements affect other instruments in your system as well as the Oscilloscope. Use system programming statements when you want to control these instruments together with the Oscilloscope. Instrument statements only affect the Oscilloscope.

---

#### NOTE

*If the programming statements in this chapter fail to execute, you may have a program error. Refer to Chapter 5 in the System Owner's Guide, which discusses error handling methods. Appendix C of this guide lists the error messages that apply to the Oscilloscope.*

---

## How Statements Control the Oscilloscope

The PC Instruments Programming Library for BASIC provides a means of controlling all PC instruments via your BASIC application program. The Library contains a number of subprograms that you call from your program. This allows you to apply your Oscilloscope, together with the other instruments, to automated measurements or tests.

When you wish to use your Oscilloscope for waveform measurements, follow the steps shown in Figure 5-1 as part of your BASIC program.

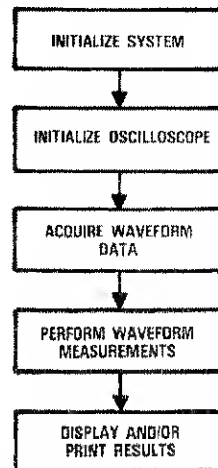


Figure 5-1. Programming Flow Chart



Each step in this flow chart corresponds to a set of programming statements. The `INITIALIZE.SYSTEM` statement is covered in the *System Owner's Guide*. The rest of the steps contain Oscilloscope-specific statements covered in this chapter and Appendix A.

It is important that you scale (set up) the Oscilloscope before data acquisition can be started. Scaling must be done when you initialize the Oscilloscope. Once the Oscilloscope is scaled, you may then acquire waveform data. If necessary or desired, you may rescale later in your application program. However, rescaling must precede the next data acquisition step that requires the rescaling. The waveform data acquisition step is a prerequisite for waveform measurements. Scaling, acquisition, and measurements must be done in that order. Once measurements are complete, the results are obtained on the computer display or other device by using `PRINT` statements.

---

## System Programming Statements

All of the system programming statements are described in the *System Owner's Guide*. Only the following statements directly affect the Oscilloscope:

**`INITIALIZE.SYSTEM`**(*statefile*). Where *statefile* is a string variable equal to a state filename that you created using the Soft Front Panel. This statement causes the Oscilloscope settings (i.e., channel, vertical sensitivity, sweep speed, etc.) to be set to the values specified in the state file. All other instruments in the system are also initialized. The `INITIALIZE.SYSTEM` statement should be used carefully because it affects all the settings of the Oscilloscope. Wherever it is used in your program, it will override the results of any previously issued instrument statements. Programming the Oscilloscope without using an `INITIALIZE.SYSTEM` statement sets your instrument (as well as all other instruments) to the factory default settings listed in the instrument specifications (Table 1-1).

---

#### NOTE

*The Rear Panel information contained in the State File must agree with your present hardware setup. The Rear Panel mode is explained in Chapter 4 of the System Owner's Guide.*

---

**CALL PANELS.** This statement interrupts your BASIC application program and returns operation to the Soft Front Panel. The Oscilloscope stops and displays the last waveform acquired before the call statement. If no waveform is stored, the Oscilloscope remains in the run mode.

This statement is used mainly for debugging programs. It allows you to check the settings on the Oscilloscope's front panel and the configuration information on its rear panel. If needed, you can also change the instrument's settings at this time. When you exit the Soft Front Panel, execution continues from the next statement in your BASIC program.

---

#### NOTE

*Changes made to the Oscilloscope configuration while in the Soft Front Panel will be carried into the BASIC program when it resumes. However, these changes are not saved in the HPSTATE.HPC file when you enter the Soft Front Panel from, and then return to, BASIC.*

---

---

## Programmed Instrument Control

In order to write programs that will run error free and give you a high confidence level in the accuracy of the measurement results, you need to understand the relationships between data acquisition cycles, sweep speeds, and random repetitive sampling.

The first step in making a measurement on a waveform is to acquire a data set that represents the waveform. When you are operating from the Soft Front Panel in the manual mode, the data acquired by the Oscilloscope is sent to the computer and displayed. The Oscilloscope attempts to collect an array of 251 points or dots (full data set); but, depending on the sweep speed, will not always return a full array. This does not present a problem in the manual mode because the new points (dots) are simply displayed in real time.

In the program mode, however, an incomplete data set (less than 251 points) does presents a situation that you must consider when writing your applications programs. For example, assume the waveform data set must be filled with all 251 points for waveform analysis and you have selected one of the fastest sweep speeds. You run your program and notice an extremely long acquisition time, possibly too long for an acceptable throughput for your application. For this reason, you are provided with program controls (timeout and percentage) to compensate for long acquisition times. The program control you use depends on your application and the choice is essentially a tradeoff between longer acquisition time with higher statistical accuracy and shorter acquisition time with lower statistical accuracy.

A "timeout" control "flags" the waveform data as incomplete if the specified percentage of points or number of averages is not collected at the end of the timeout period. You can specify the timeout period in seconds, or use the default timeout period (1.6 seconds). A

"percentage" control allows you to specify what percentage of the data set or the number of averages you consider sufficient for waveform analysis. Normally, 251 points comprise 100% of a data set. However, this number is reduced at the higher sweep speeds. 100% of the data set is represented by 201 data points at 20 $\mu$ s and by 101 data points at 10 $\mu$ s. If this number of points is acquired within the timeout period, the data set is considered complete for statistical analysis. Due to the random sampling technique used by the Oscilloscope, 90% of the data is collected very rapidly, with the remaining 10% being collected more slowly.

---

## Instrument Programming Statements

---

### NOTE

*In the programmed mode of operation, the computer display is blank unless it is commanded to display information by PRINT statements. If nothing appears to be happening in your program, you may then want to get more visual feedback by adding a PRINT statement after a variable is assigned a new value. Since they increase program execution time, you may then want to remove any unnecessary PRINT statements after your program is running to your satisfaction.*

---

In the following statement descriptions, the *label* parameter is the label of the Oscilloscope that you wish to program. It must be the same label that you assigned to the Oscilloscope when you saved the Program Shell from the Soft Front Panel. If you didn't assign any label, you must use the factory default (Scope.01 for the first Oscilloscope in your system). Where applicable, refer to Appendix C for a description of error messages.

---

#### NOTE

*The explanations given here describe what happens when an error condition is encountered. These explanations are valid only if your program includes error handling routines with STOP statements in the appropriate places. Refer to Appendix C for a description of programming error messages and to Chapter 5 of your System Owner's Guide for information about error handling routines.*

---

### General Control Statements

**INITIALIZE(label,statefile).** This statement is the same as INITIALIZE.SYSTEM except it causes only the specified instrument to be initialized to the values contained in statefile. Although this file could contain information about other instruments in your system, only the information that applies to the Oscilloscope will be retrieved. Programming the Oscilloscope without using an INITIALIZE statement sets your instrument to the factory default settings listed in Table 1-1 (Specifications).

---

#### NOTE

*The Rear Panel information contained in statefile must agree with your present hardware setup.*

---

#### Example:

```
1010 FILE$="STARTUP"  
1020 CALL INITIALIZE(SCOPE.01,FILE$)
```

This example sets the Oscilloscope with the label SCOPE.01 to the settings contained in the statefile STARTUP. You must have previously saved "STARTUP" from the Soft Front Panel.

**AUTOSCALE(*label*)**. This statement initiates an autoscale routine on Channel A (if no signal is found on A, it then attempts to autoscale on Channel B) by evaluating the current waveform and automatically scaling the vertical, trigger source, and timebase settings for optimum display of the waveform.

**Example:**

```
1010 CALL AUTOSCALE(SCOPE.01)
```

This example initiates the autoscale routine on SCOPE.01 when your program reaches line 1010. When a waveform is found, the settings are changed to the settings required to scale the Oscilloscope for this waveform. The vertical settings for the signals on Channel A and Channel B are scaled independently. The time base and trigger are set by the signal on Channel A. If there is no signal on Channel A, then scaling is set by the signal on Channel B. If no signal is found on either channel, the program halts and displays an error message. All other settings are set to the default values (refer to Table 1-1).

**CALIBRATE(*label*)**. This statement initiates self calibration. Upon successful completion of the calibration, the program continues to the next program step. If the Oscilloscope fails calibration, the program halts and an error message is displayed.

**Example:**

```
1010 CALL CALIBRATE(SCOPE.01)
```

**SET.TIMEOUT(*label, timeout*)**. This statement sets the time, in seconds, for data acquisition. If data acquisition is not completed within the period specified, an error is generated and your program halts. However, the data that was acquired is not lost, but stored in the waveform array (refer to "Acquisition Control Statements"). The default timeout value is 1.6 seconds.

**Example:**

```
1010 TIMEOUT = 1.0
1020 CALL SET.TIMEOUT(SCOPE.01, TIMEOUT)
```

In this example, the timeout time of SCOPE.01 is set to 1 second. If all 251 or the specified percentage of points or number of averages are not collected by one second, an error message is displayed.

**Vertical Control  
Statements**

**SET.SENSITIVITY**(*label,channel,v.sens,attenuation*). This statement selects the channel, vertical sensitivity, and probe attenuation factor. Table 5-1 lists the parameter options.

**Example:**

```
1010 CALL SET.SENSITIVITY
      (SCOPE.01,CHAN.A,R5MILLI,X1)
```

This example sets SCOPE.01 to Channel A with a sensitivity of 5 mV/division and probe scaling for a 1:1 probe.

Table 5-1. SET.SENSITIVITY Parameter Options

Name	Description	Ranges	Passed Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	Default or user-defined *	SCOPE.01 or user-defined *
<i>channel</i>	vertical channel	A B	CHAN.A CHAN.B
<i>v.sens</i>	volts/division range  <div> <p><b>Note</b> These ranges can only be used with a 1:1 probe.</p> <p><b>Note</b> These ranges can be used with either a 1:1 or 10:1 probe.</p> <p><b>Note</b> These ranges can only be used with a 10:1 probe.</p> </div>	5 mV/div 10 mV/div 20 mV/div  50 mV/div 100 mV/div 200 mV/div 500 mV/div 1 V/div 2 V/div 5 V/div  10 V/div 20 V/div 50 V/div	R5MILLI R10MILLI R20MILLI  R50MILLI R100MILLI R200MILLI R500MILLI R1 R2 R5  R10 R20 R50
<i>attenuation</i>	probe attenuation factor	1:1 10:1	X1 X10

\* Refer to Chapter 4 of the *System Owner's Guide* for limitations on user-defined names.

**SET.VERT.OFFSET**(*label,channel,offset*). This statement sets the vertical offset, in volts, for a specified channel. The variable *offset* must be assigned a value before the statement SET.VERT.OFFSET is called. Table 5-2 lists the parameter options.



**Example:**

```

1010 OFFSET = 0.0075
1020 CALL SET.VERT.OFFSET
      (SCOPE.01,CHAN.A,OFFSET)

```

In this example, the offset on Channel A of SCOPE.01 is set to 7.5 mV. If the offset is out of range, an error is generated.

**Table 5-2. SET.VERT.OFFSET Parameter Options**

Name	Description	Ranges*	Passed Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	Default or user-defined.	SCOPE01 or user-defined
<i>channel</i>	Vertical channel	A B	CHAN.A CHAN.B
<i>offset</i>	A variable assigned a real number value, in volts, for the vertical offset (position). Enter offset value in scientific or decimal notation. The plus sign is assumed for positive offset.		OFFSET
	Offset range depends on the volts/division full-scale range as follows:  40 mV to 4 V ranges 8 V to 16 V/ ranges 40 V range	(**1.5 times screen diameter) 12 volts maximum (no offset available)	

\* For Oscilloscope with a 1:1 input probe. For a 10:1 probe, multiply by 10.

\*\* One screen diameter = 8 times the V/div setting. E.g., a setting of 5 mV/div has a screen diameter =  $8 \times 5 \text{ mV} = 40 \text{ mV}$ . The resultant offset range is then  $1.5 \times 40 \text{ mV} = 60 \text{ mV}$ .

**SET.COUPLING**(*label,channel,coupling*). This statement sets the coupling for a specified channel to ac or dc. Table 5-3 lists the parameter options.

**Example:**

```
1010 CALL SET.COUPLING(SCOPE.01,CHAN.A,AC)
```

This example sets the coupling on Channel A of SCOPE.01 to ac.

**Table 5-3. SET.COUPLING Parameter Options**

<b>Name</b>	<b>Description</b>	<b>Ranges</b>	<b>Parameters</b>
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>channel</i>	Vertical channel	A B	CHAN.A CHAN.B
<i>coupling</i>	Vertical channel coupling is set for the specified channel.	ac dc	AC DC

**SET.POLARITY**(*label,channel,polarity*). This statement sets the trace polarity for a specified channel to positive or negative. Table 5-4 lists the parameter options.

**Example:**

```
1010 CALL SET.POLARITY(SCOPE.01,CHAN.A,POSITIVE)
```

This example sets the trace polarity on Channel A of SCOPE.01 to positive.

**Table 5-4. SET.POLARITY Parameter Options**

<b>Name</b>	<b>Description</b>	<b>Ranges</b>	<b>Passed Parameters</b>
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>channel</i>	Vertical channel	A B	CHAN.A CHAN.B
<i>polarity</i>	Trace polarity of specified channel is set to positive or negative.	positive negative	POSITIVE NEGATIVE

## Timebase Control Statements

**SET.SWEEPSPEED**(*label*, *s.speed*). This statement sets the sweep speed of the time base. Table 5-5 lists the parameter options.

### Example:

```
1010 CALL SET.SWEEPSPEED(SCOPE.01,R5MILLI)
```

This example sets the timebase of SCOPE.01 to 5 ms/division.

Table 5-5. SET.SWEEPSPEED Parameter Options

Name	Description	Ranges	Passed Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>s.speed</i>	Seconds/division range	10 ns/div 20 ns/div 50 ns/div 100 ns/div 200 ns/div 500 ns/div 1 $\mu$ s/div 2 $\mu$ s/div 5 $\mu$ s/div 10 $\mu$ s/div 20 $\mu$ s/div 50 $\mu$ s/div 100 $\mu$ s/div 200 $\mu$ s/div 500 $\mu$ s/div 1 ms/div 2 ms/div 5 ms/div 10 ms/div 20 ms/div 50 ms/div 100 ms/div 200 ms/div 500 ms/div	R10NANO R20NANO R50NANO R100NANO R200NANO R500NANO R1MICRO R2MICRO R5MICRO R10MICRO R20MICRO R50MICRO R100MICRO R200MICRO R500MICRO R1MILLI R2MILLI R5MILLI R10MILLI R20MILLI R50MILLI R100MILLI R200MILLI R500MILLI

**SET.DELAY(label, d.time).** This statement sets the pre- or post-trigger delay time in seconds. Variable *d.time* must be assigned a value before statement SET.DELAY is called. Table 5-6 lists the parameter options.

**Example:**

```
1010 D.TIME = -10e-6
1020 CALL SET.DELAY(SCOPE.01,D.TIME)
```

In this example a pre-trigger delay of SCOPE.01 is set to 10 microseconds. If the delay time is out of range, an error will be generated.

**Table 5-6. SET.DELAY Parameter Options**

Name	Description	Ranges	Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>d.time</i>	A variable assigned a real number value, in seconds, for pre-trigger or post-trigger delay. Enter delay in either scientific or decimal notation. The plus sign is assumed for post-trigger delay.  The delay time is dependent on the time/division range.	-0.5 to 250 times timebase full scale (or the screen diameter).*	D.TIME

\* One screen diameter = 10 times the ns/div setting. E.g., a setting of 20 ns/div has a screen diameter = 10 x 20 ns = 200 ns. The resultant delay range is then from -100 (0.5 x 200 ns) to + 50000 (250 x 200) ns.

## Trigger Control Statements

**SET.TRIG.SOURCE(*label*, *source*).** This statement sets the trigger source to Channel A, Channel B, or External. Table 5-7 lists the parameter options.

### Example:

```
1010 CALL SET.TRIG.SOURCE(SCOPE.01,CHAN.B)
```

This example sets the trigger source of SCOPE.01 to Channel B.

Table 5-7. SET.TRIG.SOURCE Parameter Options

Name	Description	Ranges	Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>source</i>	Trigger source	A B External	CHAN A CHAN.B EXTERNAL

**SET.TRIG.SLOPE(*label*, *slope*).** This statement sets the trigger slope to positive or negative. The trigger source must be either Channel A or Channel B. Refer to Table 5-8 for parameter options.

### Example:

```
1010 CALL SET.TRIG.SLOPE(SCOPE.01,POSITIVE)
```

This example sets the trigger slope of SCOPE.01 to positive.

Table 5-8. SET.TRIG.SLOPE Parameter Options

Name	Description	Ranges	Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>slope</i>	Trigger slope	positive negative	POSITIVE NEGATIVE

**SET.TRIG.LEVEL(*label*, *level*)**. This statement sets the trigger level in volts. The variable *level* must be assigned a value before the statement SET.TRIG.LEVEL is called. Table 5-9 lists the parameter options.

---

**NOTE**

*You can set the trigger level only if the trigger source is Channel A or Channel B and the mode is Triggered or Autotrigger.*

---

**Example:**

```
1010 LEVEL = 2.5
1020 CALL SET.TRIG.LEVEL(SCOPE.01,LEVEL)
```

This example sets the trigger level of SCOPE.01 to 2.5 volts.

**Table 5-9. SET.TRIG.LEVEL Parameter Options**

Name	Description	Ranges	Parameters
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	default or user-defined	SCOPE.01 or user-defined
<i>level</i>	A variable assigned a real number value, in volts, for the trigger level. Enter the level value in either scientific or decimal notation. The plus sign is assumed for a positive trigger level.	(2 times the screen diameter**)	LEVEL

\* For Oscilloscope with a 1:1 input probe. For a 10:1 probe, multiply the range by 10.

\*\* One screen diameter = 8 times the V/div setting. E.g., a setting of 5 mV/div has a screen diameter = 8 x 5 mV = 40 mV. The resultant trigger range is then 2 x 40 mV = 80 mV.

**SET.TRIG.MODE**(*label*, *mode*). This statement sets the trigger mode. Table 5-10 lists the parameter options.

**Example:**

```
1010 CALL SET.TRIG.MODE(SCOPE.01,TRIGGERED)
```

In this example the trigger mode of SCOPE.01 is set to triggered. When AUTO.LEVEL is used, the trigger level is set by hardware and overrides any level previously set by your program. When AUTO.TRIG is used, the Oscilloscope seeks a signal that satisfies the slope and level previously set. If this signal is not found, the Oscilloscope generates a trigger in hardware.

**Table 5-10. SET.TRIG.MODE Parameter Options**

Name	Description	Ranges	Parameters
<i>label</i>	Name e.g., SCOPE.01 or user defined.	default or user-defined	SCOPE.01 or user-defined
<i>mode</i>	Trigger mode	triggered auto-trigger auto-level	TRIGGERED AUTO.TRIG AUTO.LEVEL

---

**NOTE**

*Auto-level (AUTO.LEVEL) cannot be used if the trigger source (SET.TRIG.SOURCE) parameter is external (EXTERNAL)*

---

**Acquisition Control  
Statements**

The Oscilloscope must be configured, or "initialized", before acquiring data. After you have configured the Oscilloscope (by using the previously described statements), you can start waveform acquisition.



You can use either of two data acquisition modes, Standard or Averaged. Use the Standard mode to specify the percentage of points (251 points = 100%) that you consider adequate for analysis. In this mode, specify an integer number from 0 to 100 for the desired percentage. Use the Averaged mode when you want to specify the number of averages that you consider adequate for analysis. In this mode, you specify an integer number expressed as a power of 2, from 1 to 128. However, if you specify a number that is not a power of 2, the computer will round to the nearest power of two. For example, if you specify 120 averages, your value will be rounded to 128; 90 averages will be rounded to 64.

It is important to set the timeout control to allow the specified number of points or averages to be collected before timeout occurs (refer to "Programmed Instrument Control", previously in this chapter). If timeout occurs before data collection is completed, whatever data has been collected is stored in the waveform array. In order to obtain all the data, you can either increase the timeout time (preferable), reduce the percentage of points or number of averages. The tradeoff is accuracy (more data points) versus throughput (less collection time). However, since data is collected from the Oscilloscope only several times a second, it takes approximately 25 seconds (or longer, depending on the sweep speed) for 128 averages to be collected.

There are two data acquisition (get waveform) statements. The first, GET.SINGLE.WF, collects data for a single waveform and stores the result in integer array *wfarray%*. The second statement, GET.TWO.WF, collects data for both channels and stores the Channel A waveform data in integer array *wfarray1%* and the Channel B waveform data in integer array *wfarray2%*.

In addition to waveform data, scale factors (such as vertical sensitivity, timebase, offset, delay, etc.) needed for waveform analysis are also collected. The scale factors are then passed with the waveform data when a waveform analysis (such as risetime, frequency, etc.) is performed.

---

#### NOTE

*You must use one of the data acquisition statements described here to get waveform data and scale factors before attempting to do any waveform analysis.*

---

**GET.SINGLE.WF**(*label,channel,wfarray%(0),scalearray(0),mode,mode.spec*). This statement collects waveform data until the data set is complete or until the end of the specified timeout period. Set the timeout period with the SET.TIMEOUT command (or the default value will be used). The waveform data is stored in *wfarray%* and the scale factors are stored in *scalearray*. The complete data set depends on both the *mode* and the *mode.spec* parameters you specify in your statement (Refer to Table 5-11).

---

#### NOTE

*Whenever timeout occurs before data collection is completed, the collected data is stored in *wfarray%*.*

---

**Standard Mode Statement.** The following statements provide an example of the Standard mode of data collection.

**Example:**

```
1010 DIM WF%(251),SCALE(8)
.
.
.
1050 PERCENT = 90
1060 CALL GET.SINGLE.WF(SCOPE.01,
    CHAN.A,WF%(0),SCALE(0),STANDARD,
    PERCENT)
```

This example acquires 90% (approximately 226 points) of the waveform data from Channel A of SCOPE.01, assuming the timeout period does not end before all 226 bytes are acquired. The waveform data is stored in an integer array (*WF%*) and the scale factors are stored in a real number array (*SCALE*). Line 1010 dimensions each of the arrays used in line 1060. Line 1050 assigns the integer value 90 (90%) to the variable, *PERCENT*.

**Averaged Mode Statement.** The following statements provide an example of the Averaged mode of data collection.

**Example:**

```
1010 DIM WF%(251),SCALE(8)
.
.
.
1050 AVE.NO = 4
1060 CALL GET.SINGLE.WF(SCOPE.01,
    CHAN.A,WF%(0),SCALE(0),AVERAGE,AVE.NO)
```

This example acquires waveform data from channel A of SCOPE.01 until 4 averages are collected, assuming the timeout period does not end before data collection is complete. The waveform data is stored in an integer array (*WF%*) and the scale factors are stored in a real number array (*SCALE*). Line 1010 dimensions each of the arrays used in line 1060. Line 1050 assigns the integer value 4 (4 averages) to the variable, *AVE.NO*.

The waveform data is a quantitative representation of the waveform voltages. The integer values returned will be either 0 or a value between 8 and 248, where 0 means no data collected and 8 to 248 represents valid data. Data that is out of range is clipped at the integer values of 8 and 248.

**Table 5-11. GET.SINGLE.WF Parameter Options**

<b>Name</b>	<b>Description</b>	<b>Ranges*</b>	<b>Passed Parameters</b>
<i>label</i>	Name e.g., SCOPE.01 or user-defined.	Default or user-defined	SCOPE.01 or user-defined
<i>channel</i>	Vertical channel	A B	CHAN.A CHAN.B
<i>wfarray</i>	Integer array where the waveform data is stored. Must be dimensioned before being called.	User-defined. Must include % symbol	WF%(0) or user-defined
<i>scalearray</i>	Real number array where the scale factors are stored. Must be dimensioned before being called.	user-defined	SCALE(0) or user-defined
<i>mode</i>	Entry that specifies the Standard or Averaged mode of data collection.	standard or averaged	STANDARD or AVERAGE
<i>mode.spec</i>	Real number variable that is assigned a value equivalent to: percentage of a full data set in the Standard mode or number of averages in the Averaged mode.	0 to 100  1 to 128 (powers of 2)	PERCENT or user-defined  AVE.NO or user-defined

**GET.TWO.WF**(*label, wffarray1%(0), scalearray1(0), wffarray2%(0), scalearray2(0), mode, mode.spec*). This statement acquires waveform data from both Channel A and Channel B. Channel A data is stored in *wffarray1%* and scaling stored in *scalearray1*. Channel B data is stored in *wffarray2%* and scaling stored in *scalearray2*. This statement uses statement options similar to those of **GET.SINGLE.WF** (refer to Table 5-11). However, you must be careful not to duplicate names of previously assigned and dimensioned arrays.

### **Data Structures**

The waveform and scale factor arrays returned by both acquisition statements have the structure shown in Table 5-12.

Table 5-12. Structure of Arrays Returned by GET.SINGLE.WF and GET.TWO.WF Statements

Array	Description	
<i>wf%</i>	251-element integer array of Oscilloscope data that has been returned. The values are from 8 to 248, and zero. Zero values represent holes, or time "buckets" where data was not collected by the random sampling of the Oscilloscope.	
	<b>Subscript</b>	<b>Content</b>
	0	1st data value
	1	.
	2	.
	.	.
	.	.
	249	.
	250	251st data value
<i>scale</i>	An array of 8 real values, calculated in the Get-Waveform routine, which describe the vital Oscilloscope settings that determine the actual values in volts and seconds.	
	<b>Subscript</b>	<b>Content</b>
	0	internal Oscilloscope system flag
	1	Y-axis scale (v/div)
	2	X-axis scale (s/div)
	3	vertical offset (volts)
	4	delay time (seconds)
	5	probe attenuation (x1 or x10)
	6	Y-axis increment (v/quantization level)
	7	X-axis increment (s/sample point)

## Utility GET Statements

You can read the Oscilloscope configurations to obtain another type of data collection. This is particularly useful when the Oscilloscope configurations have been set by autoscale instead of through your SET commands. When you use the AUTOSCALE command in the programmed mode, you do not know what the configurations are until the following GET statements are executed by the computer. It may be good practice to follow each AUTOSCALE command with these GET commands in order to obtain the configurations defined by autoscale. Remember, the computer must be given a PRINT command before it displays the data returned to a variable.

**GET.VERT.INFO**(*label,channel,v.div,offset,coupling,polarity,probe*). This statement queries the data base and returns the vertical sensitivity, offset, coupling, polarity, and probe settings of the specified Oscilloscope and channel. Sensitivity is returned as a real number representing the V/div setting and offset is returned as a real number representing volts. Coupling, polarity, and probe attenuation are returned as real numbers encoded as follows:

coupling: 0 = ac, 1 = dc  
polarity: 0 = negative, 1 = positive  
probe attenuation: 1 =  $\times 1$ , 10 =  $\times 10$

### Example:

```
1070 CALL GET.VERT.INFO(SCOPE.01,CHAN.A,  
    V.DIV, OFFSET, COUPLING, POLARITY,  
    PROBE)  
1080 PRINT "VERTICAL SENSITIVITY="V.DIV  
1090 PRINT "OFFSET="OFFSET  
1100 PRINT "COUPLING="COUPLING  
1110 PRINT "POLARITY="POLARITY  
1120 PRINT "PROBE ATTENUATION="PROBE
```

In this example, the Channel A vertical scaling of SCOPE.01 is returned as a real integer number and displayed by the PRINT statements.

**GET.TIMEBASE.INFO**(*label,s.div,delay*). This statement queries the data base and returns the timebase range and time delay. The timebase range is returned as a real number equal to the s/div setting ( c.g., 10 ms/div is returned as .01). The delay is returned as a real number in seconds.

**Example:**

```
1070 CALL GET.TIMEBASE.INFO(SCOPE.01, S.DIV,  
      DELAY)
```

This example returns the sweep speed and delay scaling.

**GET.TRIG.INFO**(*label,source,level,slope,trigmode*). This statement queries the data base and returns information about the trigger source, slope, level, and mode. If the source is External, only the source and mode information is valid; the other values are returned as invalid (99). The source information is encoded as follows:

<b>source</b>	0 = Channel A	<b>mode</b>	0=triggered
			1=auto-trigger
	1 = Channel B		
	2 = External		

If the source is either Channel A or B, the slope and mode values are encoded as follows:

<b>slope</b>	0 = negative	<b>mode</b>	0 = triggered
	1 = positive		1 = auto-trigger
			2 = auto-level

*Level* is returned as a real number in volts.



## Measurement Statements

Once the waveform has been acquired, voltage, time, and statistical calculations may be performed on the data. We will now describe the CALL statements that perform these calculations. *Before calling any of these statements for the first time, be sure that all the statement variables have been previously initialized.*

---

### NOTE

*If any of the following measurement statements has parameter variables not already initialized or dimensioned, then you must do so before the statement is called in your program.*

---

**Voltage.** Voltage calculations are programmed as follows:

**CALC.WFVOLT**(label,scalearray(0),v1,v2)

This statement calls a routine that converts a waveform quantization level (v1) to an actual voltage (v2). The routine uses specified scale factors collected by the GET.WF routines.

To PRINT the returned array, you may use a routine such as:

```
1010 CALL GET.SINGLE.WF(SCOPE,01,CHAN.A,  
    WF%(0),SCALE(0),STANDARD,PERCENT)  
1020 N2=250: V2=0  
1030 FOR I=0 TO N2  
1040 V1=WF%(I)  
1050 CALL CALC.WFVOLT  
    (SCOPE,01,CHAN.A,SCALE(0), V1,V2)  
1060 PRINT V2  
1070 NEXT I
```

**Time.** Time calculations are programmed as follows:

**CALC.WFTIME**(label,scalearray(0),t1,t2)

This statement calls a routine that converts a sample point number (t1) that has a value between 0 and 250 to an actual number of seconds (t2).

To PRINT the actual time values, you may use a routine such as:

```
1010 CALL GET.SINGLE.WF(SCOPE.01,CHAN.A,  
    WF%(0),SCALE(0),STANDARD,PERCENT)  
1020 N=250: T2=0  
1030 FOR I=0 TO N  
1040 CALL CALC.WFTIME(LABEL,  
    SCALEARRAY(0), I, T2)  
1050 PRINT T2  
1060 NEXT I
```

**Statistics.** Since the measurement routines cannot process waveform data with any zero values in the array, interpolation is used to fill the holes (zero values in the waveform array are filled).

---

#### NOTE

*If any "holes" are present in the original waveform, CALC.WF.STATS will fill them by linear interpolation, thereby altering the waveform sent to a subsequent analysis routine.*

---

**CALC.WF.STATS**(*label*,*wf*%(0), *yamp*(0),  
*redge*%(0),*fedge*%(0))

This statement calls a routine that performs analysis on the data in waveform array *wf*%(0) and returns 7 real number values. The values are stored in array *yamp*(0). The values stored are quantitative values (not voltages). The following data structure represents the *yamp* array:

Subscript		Content
0	.....	internal Oscilloscope system flag
1	.....	absolute minimum
2	.....	absolute maximum
3	.....	base point or statistical minimum (0%)
4	.....	top point or statistical maximum (100%)
5	.....	10% point
6	.....	50% point
7	.....	90% point

In addition to filling the *yamp* array, this statement finds the rising and falling edges of the waveform at three points of five consecutive edges. The points are at 10%, 50%, and 90% and are stored in two integer arrays. Rising edges are stored in array *redge*% and falling edges are stored in *fedge*%. If there are more than 5 edges, the extra edges are ignored. If there are less than 5 edges, the remainder of the arrays are filled with zeros. Location zero of each edge array specifies the number of edges found.

---

#### NOTE

*Before you can fill the **yamp**, **redge**%, and **fedge**% arrays, you must have dimensioned them in a preceding program statement.*

---

## YAMP, REDGE%, and FEDGE% Dimension Statements

Use the following programming statements to dimension the *yamp*, *redge%*, *fedge%* arrays.

```
1030 DIM YAMP (8)
1040 DIM REDGE% (16)
1050 DIM FEDGE% (16)
```

The following data structures represent the *redge%* and *fedge%* arrays. Each array contains 16 bytes of information about the number and location (quantization level) of the rising and falling edges of the waveform.

### a. *redge%* array

Subscript		Content
0	.....	number of rising edges
1	.....	10% point of first edge
2	.....	50% point of first edge
3	.....	90% point of first edge
4	.....	10% point of second edge
5	.....	50% point of second edge
6	.....	90% point of second edge
.		
.		
13	.....	10% point of fifth edge
14	.....	50% point of fifth edge
15	.....	90% point of fifth edge

**b. fedge% array**

Subscript		Content
0	.....	number of falling edges
1	.....	90% point of first edge
2	.....	50% point of first edge
3	.....	10% point of first edge
4	.....	90% point of second edge
5	.....	50% point of second edge
6	.....	10% point of second edge
.		
.		
.		
13	.....	90% point of fifth edge
14	.....	50% point of fifth edge
15	.....	10% point of fifth edge

By using the programming statements described thus far, your program will acquire enough waveform data and perform the requisite waveform calculations for you to do your own waveform analysis, such as risetime, period, etc. However, you can use the programming statements in Table 5-13 to perform the waveform analysis automatically.

---

**NOTE**

*The terms used in the statements conform to the definitions given in IEEE STD 194-1977 (Pulse Terms and Definitions) and STD 181-1977 (Pulse Measurement and Analysis by Objective Techniques).*

---

Before calling any of the statements listed in Table 5-13, you must have acquired the necessary data base by first calling:

<b>GET.SINGLE.WF(Parameters)</b>	Supplies waveform and scale factor data.
<b>CALC.WF.STATS(Parameters)</b>	Statistically analyzes the waveform data, fills any holes with zeroes, and produces the <i>yamp</i> , <i>redge%</i> , and <i>fedge%</i> arrays.

#### NOTE

*Each of the measurement statement routines listed in Table 5-13 checks for sufficient data and resolution for the measurement. If sufficient data and resolution are not available, an error message is generated and the returned measurement parameter is zero.*

**\* Table 5-13. Summary of Measurement Statements**

**CALC.RISETIME**(*label, wf%(0), scale(0), yamp(0), redge%(0), risetime.value*)

This statement calculates the risetime of the acquired waveform from the 10% to 90% points and returns the value expressed in seconds.

**CALC.FALLTIME**(*label, wf%(0), scale(0), yamp(0), fedge%(0), falltime.value*)

This statement calculates the falltime of the acquired waveform from the 10% to 90% points and returns the value expressed in seconds.

**CALC.PERIOD**(*label, wf%(0), scale(0), yamp(0), redge%(0), fedge%(0), period.value*)

This statement calculates the period of the acquired waveform between the 50% points and returns the value expressed in seconds.

**\*Table 5-13. Summary of Measurement Statements (Cont.)**

**CALC.FREQUENCY**(*label, wf%(0), scale(0), yamp(0), redge%(0), fedge%(0), freq.value*)

This statement calculates the frequency of the acquired waveform, using the 50% points, and returns the value in Hz.

**CALC.PLUSWIDTH**(*label, wf%(0), scale(0), yamp(0), redge%(0), fedge%(0), pwidth*)

This statement calculates the duration between pulse start time and pulse stop time (50% points on rising and falling edges in accordance with IEEE standards) and returns the value expressed in seconds.

**CALC.MINUSWIDTH**(*label, wf%(0), scale(0), yamp(0), redge%(0), fedge%(0), mwidth*)

This statement calculates the duration between pulse stop time and pulse start time and returns the value expressed in seconds.

**CALC.OVERSHOOT**(*label, wf%(0), scale(0), yamp(0), redge%(0), fedge%(0), overshoot*)

This statement calculates the overshoot (distortion) that follows a major transition and returns the value in volts.

**CALC.PRESHOOT**(*label, wf%(0), scale1(0), yamp(0), redge%(0), fedge%(0), preshoot*)

This statement calculates the preshoot (distortion) that precedes a major transition and returns the value in volts.

**CALC.PK.TO.PK**(*label, scale(0), yamp(0), pk.to.pk.value*)

This statement calculates the absolute value of the algebraic difference between the positive peak magnitude and negative peak magnitude.

\* Before calling any of these statements, be sure to initialize the corresponding return variable (*risetime.value, falltime.value, period.value, freq.value, pwidth, mwidth, overshoot, preshoot, or pk.to.pk.value*).

**Example of Using a Measurement Statement.** In the following example, line 1060 calculates the frequency of the waveform data that was statistically analyzed in line 1040. The frequency is displayed when the variable FREQ.VALUE is printed in line 1070.

```
1040 CALL CALC.WF.STATS(SCOPE.01,WF%(0),  
    YAMP(0),REDGE% (0),FEDGE% (0))  
1050 FREQ.VALUE = 0  
1060 CALL CALC.FREQUENCY (SCOPE.01, WF%(0),  
    SCALE(0),YAMP(0), REDGE%(0), FEDGE%(0),  
    FREQ.VALUE)  
1070 PRINT "FREQUENCY =" FREQ.VALUE
```

---

## Example Program

The following program example uses some statements described in this chapter. Before trying this program, you must have saved both the Program Shell and a State file from the Soft Front Panel. Once you have saved the Program Shell and State files, exit the Soft Front Panel and run PCIBAS. Load the Program Shell and add the program lines. When entering the program lines, remember to use the same labels that you assigned from the Soft Front Panel. You must also use the same State file name in your program that you assigned from the Soft Front Panel. After completing all the statements, remember to save your program before you run it.

This program acquires the Probe Compensation signal and calculates the risetime. Before you run the program, connect the Oscilloscope probe to the Probe Comp test point on the Oscilloscope front panel (Refer to Chapter 4).



---

### NOTE

*This program is written for an Oscilloscope that has a 1:1 input probe. If your scope has a 10:1 probe, change line 1070 to:*

```
1070 CALL SET.SENSITIVITY(SCOPE.01, CHAN.A,  
    R50MILLI, X10)
```

---

### Example

program shell

```
1000 ! USER PROGRAM STARTS AT THIS LINE
1010 FILE$="YOURFILE"
1020 CALL INITIALIZE (SCOPE.01, FILE$)
1030 IF PCIB.ERR < > 0 THEN ERROR PCIB.BASERR
1040 DIM WF%(251),SCALE(8)
1050 DIM YAMP(8),REDGE%(16),FEDGE%(16)
1060 PRINT "SET VERTICAL SENSITIVITY TO  
    NEW VALUE"
1070 CALL SET.SENSITIVITY(SCOPE.01, CHAN.A,  
    R200MILLI, X1)
1080 IF PCIB.ERR < > 0 THEN ERROR PCIB.BASERR
1090 PRINT "GET A WAVEFORM"
1100 PERCENT = 90
1110 CALL GET.SINGLE.WF(SCOPE.01, CHAN.A,  
    WF%(0), SCALE(0), STANDARD, PERCENT)
1120 IF PCIB.ERR < > 0 THEN ERROR PCIB.BASERR
1130 CALL CALC.WF.STATS(SCOPE.01,WF%(0),  
    YAMP(0),REDGE% (0),FEDGE%(0))
1140 RISETIME.VALUE=0
1150 IF PCIB.ERR < > 0 THEN ERROR PCIB.BASERR
1160 CALL CALC.RISETIME(SCOPE.01, WF%(0),  
    SCALE(0), YAMP(0), REDGE%(0),  
    RISETIME.VALUE)
1170 IF PCIB.ERR < > 0 THEN ERROR PCIB.BASERR
1180 PRINT "THE RISETIME IS  
    =";RISETIME.VALUE;" SECONDS"
1190 END
```

### Comments

- Line 1010 — uses your State file name assigned from the Soft Front Panel
- Line 1020 — initializes the Oscilloscope (if this line is not used, the Oscilloscope is set to its default parameters)
- \* Line 1030 — allows you to determine if an error has been detected in the CALL.
- Line 1040 — dimensions waveform and scale arrays
- Line 1050 — dimensions waveform statistics arrays
- Line 1070 — sets vertical sensitivity to 200 mV/div for Probe Comp signal
- \* Line 1080 — allows you to determine if an error has been detected in the CALL.
- Line 1100 — specifies the percentage of a complete data set you want for your analysis.
- Line 1110 — acquires the Probe Comp signal. Stores the waveform data in array *WF%(0)* and the scale factors in *Scalearray(0)*.
- \* Line 1120 — allows you to determine if an error has been detected in the CALL.

### Comments (Cont.)

Line 1130 — statistically analyzes the waveform data and fills the yamp, redge and fedge arrays.

- \* Line 1150 — allows you to determine if an error has been detected in the CALL.

Line 1160 — calculates the risetime from the data in the arrays.

- \* Line 1170 — allows you to determine if an error has been detected in the CALL.

- \* Note that there is an error-checking statement after *every* CALL statement. This is very important because undetected CALL errors can cause your program to "crash". Refer to Chapter 5 of the *System Owner's Guide* for information about error handling.



# A

## Programming Statement Summary

---

The following is a summary of the programming statements that can be used to control the Digitizing Oscilloscope. You may use this summary as a reference guide for spelling and syntax of the available statements.

### System Statements

**CALL INITIALIZE.SYSTEM**(statefile)

### Instrument Statements

#### *General Instrument Statements*

**INITIALIZE**(label, statefile)

**AUTOSCALE**(label)

**CALIBRATE**(label)

**SET.TIMEOUT**(label, timeout)

#### *Vertical Control Statements*

**SET.SENSITIVITY**(label, channel, v.sens, attenuation)

**SET.VERT.OFFSET**(label, channel, offset)

**SET.COUPLING**(label, channel, coupling)

**SET.POLARITY**(label, channel, polarity)

#### *Timebase Control Statements*

**SET.SWEEPSPEED**(label, s.speed)

**SET.DELAY**(label, d.time)

#### *Trigger Control Statements*

**SET.TRIG.SOURCE**(label, source)

**SET.TRIG.SLOPE**(label, slope)

**SET.TRIG.LEVEL**(label, level)

**SET.TRIG.MODE**(label, mode)

### *Acquisition Control Statements*

**GET.SINGLE.WF**(label, channel, wf%(0), scale(0), mode,  
mode.spec)

**GET.TWO.WF**(label, wf1%(0), scale1(0), wf2%(0), scale2(0),  
mode, mode.spec)

### *Utility GET Statements*

**GET.VERT.INFO**(label, channel, v.div, offset, coupling,  
polarity, probe)

**GET.TIMEBASE.INFO**(label, s.div, delay)

**GET.TRIG.INFO**(label, source, level, slope, trigmode)

### *Measurement Statements*

**CALC.WFVOLT**(label, scale(0), v1, v2)

**CALC.WFTIME**(label, scale(0), t1, t2)

**CALC.WF.STATS**(label, wf%(0), yamp(0), redge%(0),  
fedge%(0))

**CALC.RISETIME**(label, wf%(0), scale(0), yamp(0),  
redge%(0), risetime.value)

**CALC.FALLTIME**(label, wf%(0), scale(0), yamp(0),  
fedge%(0), falltime.value)

**CALC.FREQUENCY**(label, wf%(0), scale(0), yamp(0),  
redge%(0), fedge%(0), freq.value)

**CALC.PLUSWIDTH**(label, wf%(0), scale(0), yamp(0),  
redge%(0), fedge%(0), pwidth)

**CALC.MINUSWIDTH**(label, wf%(0), scale(0), yamp(0),  
redge%(0), fedge%(0), mwidth)

**CALC.OVERSHOOT**(label, wf%(0), scale(0), yamp(0),  
redge%(0), fedge%(0), overshoot)

**CALC.PRESHOOT**(label, wf%(0), scale(0), yamp(0),  
redge%(0), fedge%(0), preshoot)

**CALC.PK.TO.PK**(label, scale(0), yamp(0), pk.to.pk.value)

# B

## Instrument Verification and Calibration Procedures

### Introduction

Verification and calibration procedures for your Oscilloscope are included with your PC Instruments System software. Instructions on how to load and run these procedures are given in Appendix B of your *System Owner's Guide*. When you run the verification program, step-by-step instructions appear on your computer's display to guide you through each procedure. This appendix specifies the required test equipment and briefly describes the tests performed on the instrument.

### Equipment Required

You must have an installed PC Instruments Interface Card and a PC Instruments Digitizing Oscilloscope. In addition, the equipment listed in Table B-1 is required to test the operation of the Oscilloscope.

**Table B-1. Required Equipment**

Equipment	Characteristics
Function Generator	Frequency: 1 kHz to 50 MHz Amplitude: 15 mV to 1 V/p-p Accuracy: $\pm 10\%$ Sine and square wave
DC Source	Range: $\pm 200$ mV to $\pm 4$ V Accuracy: $\pm 0.3\%$
RF Voltmeter	Accuracy: $\pm 3\%$ at 50 MHz
BNC Tee Connector	1 male, 2 female
Cables	2, Coaxial, 50-ohm

---

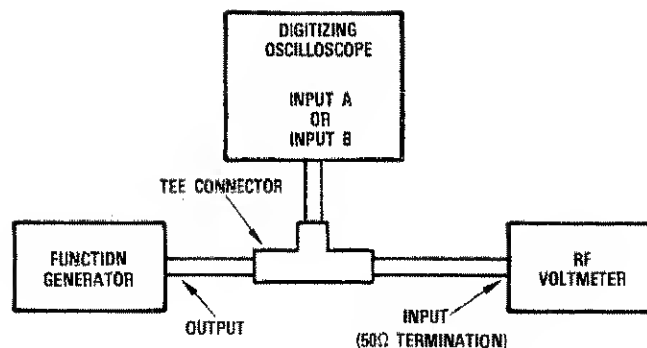
## What the Verification Test Does

When you first run the verification program, it does a system level verification that partially tests all the instruments in your system. This part of the test is described in Appendix B of your *System Owner's Guide*. After your Oscilloscope passes these preliminary tests it will appear in a menu as "SCOPE" along with the other instruments in your system. This menu is the starting point for the instrument-specific verification tests for all of your instruments. To continue testing your Oscilloscope, you must select SCOPE from this list and select f7 (TEST). When you select an instrument from the list, the ACTIVE indicator on the front of the instrument will light.

Connect the test equipment as instructed by the program. The instrument-specific verification for the Oscilloscope calibrates and tests the triggering and vertical accuracy. The triggering and vertical tests should be performed on both Channel A and Channel B of the Oscilloscope. As each test is completed, a message appears on the screen to inform you that the Oscilloscope either passed or failed that particular test.

- |                      |   |
|----------------------|---|
| <b>Calibration</b>   | This part of the verification program initiates a "self-calibration" and then checks if the resulting calibration is successful.  |
| <b>Trigger Test</b>  | This test checks the internal and external triggering capabilities of the Oscilloscope. The trigger offset, gain, and sensitivity are tested.   |
| <b>Vertical Test</b> | This test verifies that the input signal is correctly digitized by the Oscilloscope. Ac offset, dc offset, gain and bandwidth are tested at each input channel. The measured signal is compared to the expected input signal. Therefore, the accuracy of the input test depends on the accuracy of the chosen signal source. When testing the bandwidth of the Oscilloscope, the program will instruct you to connect your test equipment as shown in Figure B-1. |





**Figure B-1. Bandwidth Test Setup**

## Test Results

It is important that you complete all verification tests on both system and instrument level. If your Oscilloscope fails either the trigger or the vertical test, calibrate it and repeat the tests. If your Oscilloscope still fails the trigger or vertical test, or if you encounter a calibration error, consult your **PC Instruments Support Guide** for information on the PC Instruments exchange program.



# C

## Error Messages

### Introduction

Table C-1 lists the programming error messages that apply to the Digitizing Oscilloscope. Table C-2 lists the Soft Front Panel (SFP) errors. When programming your Oscilloscope from BASIC, the programming error messages are returned only if you use the error handling routine described in Chapter 5 of the *System Owner's Guide*.

Besides error messages, the Oscilloscope displays status messages in the field below the waveform display, just above the Scopekeys. These status messages, which are listed in Table C-3, do not always indicate an error condition.

Table C-1. Programming (HPERR.HPE) Error Messages

HPERR.HPE#	Message	Corrective Action
701	Channel must be CHAN.A or CHAN.B	Set the channel to CHAN.A or CHAN.B in the BASIC program.
702	Coupling must be AC or DC	Set coupling to AC or DC.
703	Invalid vertical sensitivity setting	Set vertical sensitivity to one of the 13 choices listed for the SET.SENSITIVITY command in Chapter 5.
704	Probe setting must be x1 or x10	Set probe x1 or x10 in the BASIC program.
705	Polarity must be POSITIVE or NEGATIVE	Set waveform polarity to POSITIVE or NEGATIVE.

**Table C-1. Programming (HPERR.HPE) Error Messages (cont.)**

HPERR.HPE#	Message	Corrective Action
706	Invalid timebase setting	Set timebase setting to one of the 24 choices listed for SET.SWEEPSPEED command in Chapter 5.
707	Source must be CHAN.A, CHAN.B, EXTERNAL	Set the trigger source to CHAN.A, CHAN.B, or EXTERNAL in the BASIC program.
708	Trig slope must be POSITIVE or NEGATIVE	Set trigger slope to POSITIVE or NEGATIVE in the BASIC program.
709	Invalid mode for trigger source	Set the trigger mode only if the source is CHAN.A or CHAN.B. If source is EXTERNAL, the mode can be TRIGGERED or AUTO.TRIG.
710	Trig level error; check source or mode	Set the trigger level only if the source is CHAN.A or CHAN.B and if mode is TRIGGERED or AUTO.TRIG. For EXTERNAL or AUTO.LEVEL, the system sets the trigger level.
711	Source must be CHAN.A or CHAN.B	Set the trigger slope only if the source is CHAN.A or CHAN.B

**Table C-1. Programming (HPERR,HPE) Error Messages (cont.)**

HPERR,HPE#	Message	Corrective Action
712	Invalid trigger mode setting	Set the trigger mode to TRIGGERED, AUTO.TRIG, or AUTO.LEVEL in the BASIC program.
713	Collection mode = AVERAGE, STANDARD only	Set the collection mode for GET.SINGLE.WF or GET.TWO.WF to either STANDARD or to AVERAGE, in the BASIC program.
714	Percent of completion must be 0-100	Set the percent of completion for collection in the STANDARD mode to a value from 0 to 100.
715	No. of aver 1,2,4,8,16,32, 64,128 only	Set the number of averages for collection in the AVERAGE mode to 1,2,4,8,16,32,64, or 128. Values greater than 128 or negative values will return this error message. Values not expressed in powers of two will be decremented to nearest power of two and no error returned.

**Table C-1. Programming (HPERR,HPE) Error Messages (cont.)**

HPERR,HPE#	Message	Corrective Action
716	Timeout must be a positive real number	Set timeout value for collection of data from the Oscilloscope to a positive real number that represents seconds.
717	Offset is out of range	Set the offset to within the range limit as specified in Chapter 5 for SET.VERT.OFFSET command.
718	Trigger level is out of range	Set the trigger level to within the range limit as specified in Chapter 5 for SET.TRIG.LEVEL command.
719	Timebase delay is out of range	Set the timebase delay to within the limits specified in Chapter 5 for SET.DELAY command.
720	Calibration failed	Calibration of the Oscilloscope has failed. No corrective action can be taken. Refer to PC Instruments Support Guide.

Table C-1. Programming (HPERR.HPE) Error Messages (cont.)

HPERR.HPE#	Message	Corrective Action
721	Timeout has occurred on data collection	The set (or 1.6-second default) timeout value was exceeded during the data collection routine done in response to a GET.WF command. Increase the timeout value (use SET.TIMEOUT) or decrease the number of averages.
722	Insufficient data for measurement	Not enough data collected for an accurate measurement. Check that the vertical and horizontal settings are appropriate for the input waveform.
723	No scope response	The Oscilloscope is not responding in the expected time. Check power cord, cables, and all connections.
724	Scope not triggered	The Oscilloscope was not triggered during a data acquisition.
725	Cal factor number must be 0-13	No corrective action can be taken. Oscilloscope requires service. Refer to <i>PC Instruments Support Guide</i> .

**Table C-1. Programming (HPERR.HPE) Error Messages (cont.)**

HPERR.HPE#	Message	Corrective Action
726	Setting is not within resolution limit	Offset, trigger level, and delay settings are less than the resolution of the Oscilloscope.
727	Offset & triglevel out of range	Both the OFFSET and TRIGLEVEL are now invalid because of a new v.div setting.
728	Autoscale failed	Autoscale mode failed to find a signal.



Table C-2. Soft Front Panel Error Messages

Error Message	Corrective Action
Both channels are off	Markers cannot be set because neither channel is ON for referencing.
Channel is off	Marker cannot be set on a specified channel when it is OFF.
Markers are not on the same channel	Autopositioning of markers cannot be done when markers are referenced to different channels.
No corresponding edge is found	Time markers cannot be auto positioned on specified edge because it does not exist (change the sweep speed).
Delay reset in range	The specified time delay was set out of range, but has been reset to the closest in-range value.
Offset reset in range	The specified vertical offset was set out of range, but has been reset to the closest in-range value.
Triglevel reset in range	Specified trigger level was set out of range, but has been reset to the closest in-range value.
Offset & Triglevel reset in range	The offset and the trigger level were set out of range for the specified vertical sensitivity setting. Both values have been reset to be in range.
Insufficient data for measurements	Not enough waveform data exists or existing data is not of sufficient magnitude to successfully analyze waveform and perform valid measurements.
No saved data in wfmgmn record	You requested recall of a waveform, but no data was previously saved in the specified record via a SAVE command.

**Table C-2. Soft Front Panel Error Messages (cont.)**

Error Message	Corrective Action
No signal found	Unsuccessful autoscale; check Oscilloscope connections and input signal.
No response from scope	Attempt to start Oscilloscope or collect data was unsuccessful. Check connections and input signal.
Running	Oscilloscope is now collecting data.
Trig auto level search failed	Oscilloscope is in auto level mode but has failed to find a trigger.
Calibration failed	Check your connections. If they are correct, instrument requires service. Refer to the <i>PC Instruments Support Guide</i> .

## Soft Front Panel (SFP) Note Messages

When the oscilloscope is the active instrument, its NOTE window is continually updated to display the Oscilloscope's status. The following messages, which are not necessarily errors, keep you informed of the oscilloscope's activity.

**Table C-3. Soft Front Panel Note Messages**

SFP Message	Message Information
no signal	No signal is being received; check Oscilloscope connections and input signal.
running	Oscilloscope is running and collecting data for display; if no waveform is seen, check offset and coupling.
stopped	You have stopped the Oscilloscope. To restart, select the RUN Scopekey.
single	You have specified the SINGLE mode by selecting the SINGLE Scopekey. To restart, either select the RUN Scopekey or select SINGLE again to collect another data set.
no scope response	Oscilloscope is not accepting set up commands; check wiring connections.
searching for trigger	Oscilloscope is in the AUTO LEVEL trigger mode and searching for trigger. View the TRIG MENU to see the trigger level is and how the search is proceeding.
awaiting trigger	Oscilloscope is in TRIGGERED mode and has not been able to find a trigger.

**Table C-3. Soft Front Panel Note Messages (cont.)**

SFP Message	Message Information
auto triggering	Oscilloscope is in AUTO TRIGGERING mode. It has not been able to find a trigger and is self-triggering every 25 ms.
trig level at max limit	Oscilloscope is in AUTO LEVEL mode and has reached the maximum limit without finding a trigger. Try changing the vertical sensitivity setting or checking the input signal.
trig level at min limit	Oscilloscope is in AUTO LEVEL mode and has reached the minimum limit without finding a trigger. Try changing the vertical sensitivity setting or checking the input signal.